Subject: Simulation Requirements Document
Posted by dhiman on Thu, 27 May 2004 15:31:51 GMT
View Forum Message <> Reply to Message

Friends,
The Worldwide LC simulation working group is preparing a document that lists the requirements for a common detector simulation program. A preliminary draft of this document is ready for your viewing. Please see message (#98) posted today on the Common Simulation Framework forum.

On behalf of the Worldwide LC Simulation Working Group,
Dhiman Chakraborty
NIU/NICADD

Subject: Protoype Simulation - Cell Numbering and Coordinate Frames
Posted by musat on Wed, 09 Jun 2004 15:04:48 GMT
View Forum Message <> Reply to Message

Hi,

It was suggested previously to move this discussion to the full simulation forum.

Here at the LLR, we wrote some suggestions concerning the coordinate system and the cell numbering. The document is

http://polywww.in2p3.fr/~musat/Proto/Ecal.html

I'm currently writing the Ecal prototype implementation and would like to make a new Mokka release this month, so perhaps we could try to find an (at least first) agreement on these points in a delay of, say, ten days.

What do you think about that?

Cheers,

Gabriel

========== ===
Roman wrote:

Dear Colleagues,

I would like to pick up the discussion on coordinate frames again. First of all thanks to all of you for contributing to the discussion
and sorry that I didn't react earlier to all the E-Mails.

I guess we have agreed ...

- to align the detectors along the z-axis. To other items
 see my answers to Davids E-Mail below

- to try and find an as far as possible common index scheme to be packed into one LCIO word

A good time to settle the discussion is the Calice-Meeting by the end of June. Should we strive for a written proposal to
be presented to the other Calice members ? If yes, who's going to write it (Gabriel was proposed, I would volounteer, too) ?
I am quite sure that any proposal will trigger a lively discussion at CERN .
Once the conventions are fixed it might be good to determine a time when
all the outcome has to implemented into Mokka.

Looking forward to your comments.

Cheers,

Roman

David Ward wrote:

> Hi Roman,
>
> Sorry, I should have replied sooner. A few comments below...
>
> On Thu, 6 May 2004, Roman Poeschl wrote:
>
>
>
>> Dear Collegue,
>>
>> over the last few weeks people at NIU and DESY have been working on the
>> implementation of the hadronic tile calorimeter into the MOKKA simulation
>> framework. A number of points have come up which should be - we believe -
>> coordinated with the other groups doing simulations for the test
>> calorimeters.

>>
>> The main issues are
>> - a common coordinate system
>> - (possibly) a central cell numbering scheme for the prototype.
>>
>> We are aware of at least two different coordinate systems currently in
>> use. Obviously we all would profit very much from a common system. I like
>> to propose the following convention:
>>
>
>
> I agree we should harmonise things, and try to settle it once and for all.
>
>
>
>> * the z-axis is defined as the direction of the beam.
>>
>
>
> This seems the most natural choice, though the LLR-defined prototype in
> Mokka uses y.
>
>
>> * We use a standard right handed coordinate system
>> * the origin of the system is the symmetry point of the calorimeter in x,y
>> * the z=0 point is at the face of the ECAL.
>>
>>
>
>
> For z=0, what exactly do you mean by the face? You mean the first layer
> of any material in the ECAL (epoxy, probably)?
>
>

Yes, I mean the first layer of any material. The z-Position of the first layer in the ECAL would then be
z_1stlay = 0 - (Layer-Dimension)/2. or similar.

In the HCAL we currently have put the z-position of the layer into the middle of the scintillating tiles.

> For (x,y) I assume you mean the symmetry point of the HCAL? The ECAL is a
> bit more complicated because of the staggering of layers of Si, but if we
> specify (x,y)=(0,0) in terms of the HCAL, the ECAL just has to be aligned
> w.r.t. the HCAL.
>
>
Yes, I mean the symmetry point of the HCAL.

>
>
>
>> To make a fast access to the cell possible a cell numbering
>> scheme is needed.
>> We propose a simple numbering scheme, based on cells of 1x1cm2 for the
>> HCAL. The numbering starts at the lower left corner of each layer.
>> * i: row of cell
>> * j: column of cell
>> * k: layer number of cell
>> We propose to start the numbering of layers new for each subdector type.
>> (ECAL, HCAL, Tailcatcher). The three indices should be packed into
>> one word for output.
>>
>
>
> Starting at 0 in each case, or 1?
>
>
I would always start with 0.

> The existing ECAL scheme has two levels, the wafer indices and the pad indices within each wafer. I assume you are proposing a single index in each direction instead. That's probably OK, I just wanted to be clear.
>
>
Yes, I would prefer this.

Maybe the ECAL experts can work out a numbering scheme which is oriented on what was discussed so
far and which is best for their needs.

> I'm not quite clear about the 1x1 cm cells. These are meant to be virtual

> cells, I assume, at least in the HCAL. In the ECAL how do these map onto
> the physical pads (because of gaps between wafers, staggers between layers
> etc). Just need to clarify this. Maybe in the ECAL they are just the
> physical pads. How about the HCAL - do 1x1 cm cells map exactly onto your
> complicated pattern of different-sized cells? Are there issues relating
> to gaps between tiles, of are the gaps always exactly on a 1 cm grid?
>
>
We can map the 1x1 virtual cells onto the geometry of the 'real'-HCAL. The cell dimensions of the real HCAL are integer multiples of the 1x1cm2 geometry. In first approach there will be no gaps between
the tiles in lateral direction.

>
>
>> To make exchanging and using data simpler, it would be great if
>> all groups could agree to a common index scheme and also to a
>> common packing scheme of these indices into one LCIO word.
>>
>>
>
>
> I agree.
>
> So, I have no problem with this in principle, I just think we have to make
> any proposal absolutely clear, and be sure it does all we need.
>
> Best regards,
>
>   David.
>
>
>
> +-- --------+
> | Dr. David R. Ward   University of Cambridge, |
> | E-mail: drw1@hep.phy.cam.ac.uk   Cavendish Laboratory, |
> | Phone: 44 1223 337242   (Room 939 Rutherford building) |
> | or 44 1223 335630 (College) Madingley Road, |
> | Fax: 44 1223 353920   Cambridge, CB3 0HE, U.K. |
> +-- --------+
>
>

>

Subject: Re: Protoype Simulation - Cell Numbering and Coordinate Frames
Posted by poeschl on Thu, 24 Jun 2004 12:23:53 GMT
View Forum Message <> Reply to Message

Dear Colleagues,

please note that there is also a webpage on a proposal for
a coordinate frame and a numbering scheme for the Mokka
implementation of the analogue Hcal.

Please check under

http://www-flc.desy.de/flc/science/hcal/simsoft/coord/Hcal_C oord.html

Cheers,

Roman

Subject: Protoype Simulation - Cell Numbering and Coordinate Frames
Posted by musat on Thu, 15 Jul 2004 12:51:38 GMT
View Forum Message <> Reply to Message

Hello,

People from LLR and DESY reached to a common point of view and wrote some suggestions concerning the coordinate system and the cell numbering. The document is

http://polype.in2p3.fr/geant4/tesla/www/mokka/ProtoDoc/Coord inatesAndNumbering.html

These suggestions were used in the new implementation of the Calice Ecal prototype at LLR, and are going to be used in the new Hcal implementation at DESY.

Cheers,

Gabriel MUSAT

Subject: requirements for using ILC software with full simulation
Posted by iglesias on Thu, 20 Nov 2008 11:45:03 GMT
View Forum Message <> Reply to Message

Hello:
I would like to use the ILC software to run PFA Pandora in Full Simulation. I'm from Santiago University in Spain and I'm new in the ILC project.

I try to install the ilc soft in my local computer (using the binary release from http://ilcsoft.desy.de/ilcsoft-bin-releases/v01-04-sl4-32bit -with-root.tgz) but I have incompatibility problems so now I would like to work directly with the SL4 DESY machine.

Is it possible from my departament university?
How I can access to /afs/desy.de/?

What requirements are needed?
- afs installation in my machine?
- SL4 intallation in my machine?
- machine with 32 or 64 bits?
- DESY usser account to have permission?

Anything else?

Thank you very much in advance.

Carmen

Subject: Re: requirements for using ILC software with full simulation
Posted by gaede on Thu, 20 Nov 2008 15:39:06 GMT
View Forum Message <> Reply to Message

Hi Carmen,

in order to use the afs installations at DESY all you need:
- an SL4 (32 bit system)
- afs client installed

no DESY account is needed.

Frank.

Subject: Re: requirements for using ILC software with full simulation
Posted by iglesias on Fri, 21 Nov 2008 08:55:44 GMT
View Forum Message <> Reply to Message

Hello Frank,

Thanks for your answer.

But I’m not sure about the need to have special permission to see the directories under /afs/desy.de/group/it/ilcsoft, because yesterday, when I tried with the binary release installation of ils software, I needed to download by myself the geant4 file and copy them to my local directory because I don’t have permission to see inside the directory:
/afs/desy.de/i586_rhel40/products/g4dataPhotonEvap
And I obtained the nest error message:
> /afs/desy.de/i586_rhel40/products > cd g4dataPhotonEvap
> cd: permission denied: g4dataPhotonEvap
Carmen

Subject: Re: requirements for using ILC software with full simulation
Posted by engels on Fri, 21 Nov 2008 10:34:15 GMT
View Forum Message <> Reply to Message

Hi Carmen,

I'll try to contact our IT colleagues to fix this.

Cheers,
Jan

Subject: Converter from Les Houches event xml files to stdhep format?
Posted by rcowan on Fri, 26 Feb 2010 21:36:48 GMT
View Forum Message <> Reply to Message

Hi,

Does anyone know of a converter that can read Les Houches event xml files and write out stdhep format files? The Les Houches event format is described in hep-ph/0609017.

Thanks,

--Ray

Subject: Re: Converter from Les Houches event xml files to stdhep format?
Posted by NormanGraf on Sat, 27 Feb 2010 00:34:17 GMT
View Forum Message <> Reply to Message

Hello Ray,
I'm not aware of any existing code which does what you are asking. On the other hand, I'm also not aware of any standard event generators which write out the xxml format. What is the source of your files? If you really need the full functionality of the Les Houches file format, then that cannot be shoe-horned into the stdhep format. If you don't need any of the extensions, then you would be better off simply writing out stdhep files to begin with. Any additional information would be appreciated.
Norman

Subject: Re: Converter from Les Houches event xml files to stdhep format?
Posted by rcowan on Sat, 27 Feb 2010 01:53:43 GMT
View Forum Message <> Reply to Message

Hi Norm,

The Les Houches xml format event files that I have were generated by a theorist using the CompHEP Feynman diagram package. The goal is to use them as input to SLIC along with the preliminary compact description of our MIT-JLAB experiment.

Below I include the first several lines of xml from the top of one of the files as a sample of the format.

Thanks much,
--Ray

```
<LesHouchesEvents version="1.0">
<!-- File generated with CompHEP 4.5.1 -->
<!--
    Preliminary version, it is compatible with the Les Houches event file
    format (hep-ph/0609017), but contains extra tags.
-->
<header>
```

<?xml version="1.0" encoding="UTF-8"?>
<samples xmlns="http://mcdb.cern.ch/hepml/0.2/"
  xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
  xsi:schemaLocation="http://mcdb.cern.ch/hepml/0.2/
file:/home/dudko/MCDB/hepml/schemas/0.2/hepml.xsd">
  <files>
    <file>
      <eventsNumber> 10000</eventsNumber>
      <crossSection unit="pb" errorMinus=" 1.2628E+06" errorPlus=" 1.2628E+06">
4.7955E+07</crossSection>
      <fileSize> 7665814</fileSize>
      <checksum type="md5"></checksum>
      <comments></comments>
      <location>
        <path/>
      </location>
    </file>
  </files>
  <description>
    <title></title>
    <abstract></abstract>
    <authorComments></authorComments>
    <experimentGroup>
      <experiment></experiment>
      <group></group>
      <responsiblePerson></responsiblePerson>
      <description></description>
    </experimentGroup>
    <generator>
      <name>CompHEP</name>
      <version>4.5.1</version>
      <homepage>http://comphep.sinp.msu.ru</homepage>
      <description>
        CompHEP: a package for evaluation of Feynman diagrams, integration over
multi-particle
        phase space and event generation (supported in part by RFBR grants 96-02-19773-a,
        99-02-04011-a, 01-02-16710-a, 04-02-17448-a). CompHEP Collaboration:
        E.Boos, V.Bunichev, M.Dubinin, L.Dudko, V.Edneral, V.ILyin, A.Kryukov, V.Savrin
        (SINP MSU, Moscow, Russia), A.Semenov (JINR, Dubna, Russia), A.Sherstnev
        (SINP MSU, Moscow, Russia and University of Cambridge, UK)
      </description>
    </generator>

Subject: Converting HepMC to Mokka compatible format
Posted by Ryan_Page on Wed, 21 Apr 2010 08:20:47 GMT
View Forum Message <> Reply to Message

Hi,

I have events that I generated in pythia 8 (C++ version) that I am trying to run through Mokka. I have used HepMC IO to create *.dat files and I was hoping that someone out there knew of a method, either using 3rd party software or something that I missed in HepMC, to save the events in a format that works with Mokka.

Thanks

Ryan

Subject: Re: Converting HepMC to Mokka compatible format
Posted by NormanGraf on Wed, 21 Apr 2010 08:49:40 GMT
View Forum Message <> Reply to Message

Hello Ryan,
Is there any particular reason you need to use pythia 8? Or could you use pythia 6? Could you use any of the very large numbers of existing stdhep files?
Norman

Subject: Re: Converting HepMC to Mokka compatible format
Posted by gaede on Wed, 21 Apr 2010 08:54:24 GMT
View Forum Message <> Reply to Message

Hi,

Mokka can read either stdhep binary files or ASCII files that follow the hepevt common block (old pytha) - either in a long or short version. I don't know if pythia C++ can still write the hepevt ascii. In any case it should be straight forward to write a script that converts the HepMC ascii into one of the hepevt formats.
For details see code snippet from $MOKKA/source/Kernel/LCAscHepRdr.cc below.

-Frank.

```
   int NHEP;  // number of entries
   inputFile >> NHEP;

//...

   int ISTHEP;   // status code
   int IDHEP;    // PDG code
   int JMOHEP1;  // first mother
   int JMOHEP2;  // last mother
   int JDAHEP1;  // first daughter
   int JDAHEP2;  // last daughter
   double PHEP1; // px in GeV/c
   double PHEP2; // py in GeV/c
   double PHEP3; // pz in GeV/c
   double PHEP4; // energy in GeV
   double PHEP5; // mass in GeV/c**2
   double VHEP1; // x vertex position in mm
   double VHEP2; // y vertex position in mm
   double VHEP3; // z vertex position in mm
   double VHEP4; // production time in mm/c

   std::vector<int> *daughter1 = new std::vector<int> ();
   std::vector<int> *daughter2 =  new std::vector<int> ();

   for( int IHEP=0; IHEP<NHEP; IHEP++ )
     {
   if ( theFileFormat == HEPEvt)
     inputFile >> ISTHEP >> IDHEP >> JDAHEP1 >> JDAHEP2
        >> PHEP1 >> PHEP2 >> PHEP3 >> PHEP5;
   else
     inputFile >> ISTHEP >> IDHEP
          >> JMOHEP1 >> JMOHEP2
          >> JDAHEP1 >> JDAHEP2
          >> PHEP1 >> PHEP2 >> PHEP3
          >> PHEP4 >> PHEP5
          >> VHEP1 >> VHEP2 >> VHEP3
          >> VHEP4;
```

Subject: Re: Converting HepMC to Mokka compatible format
Posted by Ryan_Page on Wed, 21 Apr 2010 10:08:36 GMT
View Forum Message <> Reply to Message

Hi,

The only reason I choose to use pythia 8 is I am familiar with C++, so I understood what was going on and could check things easier. I am using a sthep data set (single particle test events), but I need more than 10,000 events for what I want to do and I thought it would be easy to create some more.

Subject: Re: Converting HepMC to Mokka compatible format
Posted by NormanGraf on Wed, 21 Apr 2010 15:33:26 GMT
View Forum Message <> Reply to Message

Hello Ryan,
Thanks for your response. I asked because, to the best of my knowledge, Pythia8 is not yet up to speed for general physics use. If all you need are single particle, or other such simple events, please let us know. Chances are your sample already exists or can be easily created for you.

As you may know, HepMC does not have a standard binary file format. This has been one of the main reasons slowing our adoption of this event record format. If, however, you provide the some additional details for the .dat file you wrote out,
I am sure that we can come up with a mechanism to read your events.

Norman

Subject: Re: Converting HepMC to Mokka compatible format
Posted by weuste on Mon, 26 Apr 2010 07:47:22 GMT

I had a similar problem. Based on Frank's excerpt from Mokka I wrote a hepevt output formatter. I seems to work for the long version of hepevt, but the short version (i.e. .HEPEvt) seems to crash Mokka for some reason. Maybe there is still some glitch in here, but it might still be a helpful start for you.

```
#include "Pythia.h"

#include <iostream>
#include <iomanip>

using namespace Pythia8;
using namespace std;

int main()
{
 Pythia pythia;

 pythia.readString("SLHA:file = sps1a.spc");
 pythia.readString("SUSY:all = on");
 pythia.init("ss_out.lhef");

 pythia.settings.listChanged();
 pythia.particleData.listChanged();

 int fails = 0;
 int nrAllowedFails = 10;
 int evtNr = 0;
 bool longHepEvt = true;

 ofstream out("out.hepevt");

 while (true)
 {
  if (!pythia.next())
  {
   if (pythia.info.atEndOfFile())
   {
    cout << "End of input file reached. Exiting..." << endl;
    break;
   }
```

```
   if (++fails > nrAllowedFails)
   {
    cerr << "WARNING: Number allowed fails of Pythia::next() is reached! Will exit program!" <<
endl;
    break;
   }
   cout << "WARNING: There was an error when calling Pythia::next()! Will try again." << " (" <<
setw(3) << fails << "/" << nrAllowedFails << " fails)" << endl;
   continue;
  }

  if (++evtNr % 100 == 0)
   cout << "Status: " << setw(6) << evtNr << " (" << setw(3) << fails << "/" << nrAllowedFails << "
fails)" << endl;

  out << pythia.event.size() << endl;  // NHEP, nr of particles in this event

  for (int i = 0; i < pythia.event.size(); i++)
  {

   // try new format, i.e. long hepevt format
   // see
http://forum.linearcollider.org/index.php?t=msg&th=706&rid=0&S=f812b9be4e054ce14daec834e2
b9f2c6
   // for more infos

   // the pythia status code is different from the HepEVT status
   // HepEvt: 0: ignore - 1: final state particle - 2: decayed particle, just included to preserve history
- 4+ : reserved/builder specific/user, treat as 0
   out << setw(4) << (pythia.event[i].isFinal() ? 1 : 2) << ' '; // ISTHEP, status code
   out << setw(8) << pythia.event[i].id() << ' ';     // IDHEP, PDG code
   if (longHepEvt) {
    out << setw(4) << pythia.event[i].mother1()+1 << ' ';   // JMOHEP1, first mother id
    out << setw(4) << pythia.event[i].mother2()+1 << ' ';   // JMOHEP2, last mother id
   }
   out << setw(4) << pythia.event[i].daughter1()+1 << ' ';   // JDAHEP1, first daughter id
   out << setw(4) << pythia.event[i].daughter2()+1 << ' ';   // JDAHEP2, last daughter id

   // now start the info in double, so switch the output to that format
   out.setf(ios_base::scientific);
   out.precision(8);
```

```
   out << setw(15) << pythia.event[i].px() << ' ';  // PHEP1, px in GeV/c
   out << setw(15) << pythia.event[i].py() << ' ';  // PHEP2, py in GeV/c
   out << setw(15) << pythia.event[i].pz() << ' ';  // PHEP3, pz in GeV/c
   if (longHepEvt)
    out << setw(15) << pythia.event[i].e() << ' ';  // PHEP4, energy in GeV
   out << setw(15) << pythia.event[i].m() << ' ';  // PHEP5, mass in GeV/cc

   if (longHepEvt) {
    out << setw(15) << pythia.event[i].xProd() << ' '; // VHEP1, x vertex pos in mm
    out << setw(15) << pythia.event[i].yProd() << ' ';  // VHEP2, y vertex pos in mm
    out << setw(15) << pythia.event[i].zProd() << ' ';  // VHEP3, z vertex pos in mm
    out << setw(15) << pythia.event[i].tProd() << ' ';  // VHEP4, production time in mm/c
   }

   out << endl;

  }
 }
 out.close();

 pythia.statistics();



 return 0;
}
```

Subject: Re: Converting HepMC to Mokka compatible format
Posted by NormanGraf on Thu, 29 Apr 2010 17:04:09 GMT
View Forum Message <> Reply to Message

Hello Ryan,
 Were you able to resolve your problem?
Norman

Subject: Re: Converting HepMC to Mokka compatible format
Posted by weuste on Fri, 30 Apr 2010 09:09:12 GMT
View Forum Message <> Reply to Message

There was a bug in the code.

weuste wrote on Mon, 26 April 2010 00:47...

```
...

int main()
{
...
  for (int i = 0; i < pythia.event.size(); i++)
  {

                ...

   if (longHepEvt) {
    out << setw(4) << pythia.event[i].mother1()+1 << ' ';   // JMOHEP1, first mother id
    out << setw(4) << pythia.event[i].mother2()+1 << ' ';   // JMOHEP2, last mother id
   }
   out << setw(4) << pythia.event[i].daughter1()+1 << ' ';   // JDAHEP1, first daughter id
   out << setw(4) << pythia.event[i].daughter2()+1 << ' ';   // JDAHEP2, last daughter id



...

  }
 }
...
}
```

is has to be:

```
   if (longHepEvt) {
    int JMOHEP1 = pythia.event[i].mother1() > 0 ? pythia.event[i].mother1()+1 : 0;
    int JMOHEP2 = pythia.event[i].mother2() > 0 ? pythia.event[i].mother2()+1 : 0;
    out << setw(4) << JMOHEP1 << ' ';        // JMOHEP1, first mother id
    out << setw(4) << (JMOHEP2 == 0 ? JMOHEP1 : JMOHEP2) << ' '; // JMOHEP2, last mother id
   }
```

```
int JDAHEP1 = pythia.event[i].daughter1() > 0 ? pythia.event[i].daughter1()+1 : 0;
int JDAHEP2 = pythia.event[i].daughter2() > 0 ? pythia.event[i].daughter2()+1 : 0;
out << setw(4) << JDAHEP1 << ' ';        // JDAHEP1, first daughter id
out << setw(4) << (JDAHEP2 == 0 ? JDAHEP1 : JDAHEP2) << ' '; // JDAHEP2, last daughter id
```

Subject: Re: Converting HepMC to Mokka compatible format
Posted by Ryan_Page on Fri, 30 Apr 2010 09:30:56 GMT
View Forum Message <> Reply to Message

Hi,

Thank you for the code it has proved very useful, the events are running through Mokka now without crashing!!

Thanks

Ryan

Subject: How to count events in .stdhep file
Posted by protopop on Mon, 03 Feb 2014 10:45:18 GMT
View Forum Message <> Reply to Message

There might be an obvious answer to this, but I could not find it.

Say I use this as input for my simulations, and I want to split it by making use of /generator/skipEvents:
/ilc/datasets01/350gev/hzee/gen/00001335/000/hzee_gen_1335_1 6.stdhep

Is there a simple way to see how many events are in there in the first place ?

Subject: Re: How to count events in .stdhep file
Posted by sailer on Mon, 03 Feb 2014 12:18:41 GMT
View Forum Message <> Reply to Message

Maybe there is a better way, but I use the
stdhepjob

utility from LCIO.

stdhepjob infile.stdhep out.slcio -1

This will run over all events and tell you how many were processed.

Subject: Re-running a given simulations set
Posted by protopop on Mon, 03 Feb 2014 14:09:33 GMT
View Forum Message <> Reply to Message

Thank you for the reply to my first question!

I have a few other questions now. Say I have on disk locally

/ilc/prod/clic/350gev/hzee/ILD/DST/00001338/000/hzee_dst_133 8_*.slcio

and say I want to re-simulate these events with a slightly modified ILD geometry and be able to compare with the set on DIRAC.

Looks like the corresponding "gen" set is not in the catalogue ...

/ilc/prod/clic/350gev/hzee/gen/00001338/000

I was using /ilc/prod/clic/350gev/hzee/gen/00001335/000 for the moment, and running

Mokka -U -e ./particle.tbl 1335_93_mdf1.steer

where 1335_93_mdf1.steer contains this

/Mokka/init/lcioFilename hzee_sim1335_93_mdf1.slcio
/Mokka/init/lcioWriteMode WRITE_NEW
/Mokka/init/MokkaGearFileName Gear1335_93_mdf1.xml

/Mokka/init/user consult
/Mokka/init/dbPasswd consult

/Mokka/init/detectorModel CLIC_ILD_CDR500
/Mokka/init/lorentzTransformationAngle 10 mrad
/Mokka/init/physicsListName QGSP_BERT

/Mokka/init/rangeCut 0.1 mm # Geant4 production range cut [default is 0.005 mm]

/Mokka/init/TPCCut 10 MeV # Default is 10MeV

/Mokka/init/lcioDetailedTRKHitMode VXDCollection
/Mokka/init/lcioDetailedTRKHitMode FTDCollection
/Mokka/init/lcioDetailedTRKHitMode ETDCollection
/Mokka/init/lcioDetailedTRKHitMode SETCollection
/Mokka/init/lcioDetailedTRKHitMode SITCollection
/Mokka/init/lcioDetailedTRKHitMode TPCCollection

To run with modified detector parameters use command line option '-U'
/Mokka/init/globalModelParameter Ecal_nlayers1 17
/Mokka/init/globalModelParameter Ecal_nlayers2 8
/Mokka/init/globalModelParameter Ecal_radiator_layers_set1_thickness 2.4
/Mokka/init/globalModelParameter Ecal_radiator_layers_set2_thickness 4.8
(above numbers from http://agenda.linearcollider.org/getFile.py/access?contribId
=50&sessionId=9&resId=0&materialId=slides&co nfId=5686)

this simply sets hzee_gen_1335_93.stdhep as input
/Mokka/init/initialMacroFile 1335_93_mdf1.g4macro

/Mokka/init/printLevel 1
/Mokka/init/BatchMode true

So, does anyone know exactly ...

1) How can I find the corresponding generated (.stdhep) events ?
2) Where could I get the correct Mokka settings for this set ?
3) Is the particle.tbl file needed

Subject: Re: Re-running a given simulations set
Posted by sailer on Mon, 03 Feb 2014 14:25:43 GMT
View Forum Message <> Reply to Message

protopop wrote on Mon, 03 February 2014 06:09
1) How can I find the corresponding generated (.stdhep) events ?
2) Where could I get the correct Mokka settings for this set ?
3) Is the particle.tbl file needed

1) use "ancestor" in the dirac filecatolog on the file, it will tell you were it comes from

FC:/ilc/prod/clic/350gev/hzee/ILD/DST/00001338/000>ancestor hzee_dst_1338_1.slcio -1
/ilc/prod/clic/350gev/hzee/ILD/DST/00001338/000/hzee_dst_1338_1.slcio
1 /ilc/prod/clic/350gev/hzee/ILD/SIM/00001337/000/hzee_sim_1337_126-1-500.slcio
2 /ilc/prod/clic/350gev/hzee/ILD/SIM/00001336/000/hzee_sim_1336_77.slcio
3 /ilc/prod/clic/350gev/hzee/gen/00001335/000/hzee_gen_1335_73.stdhep

the 00001338 or similar is the production ID, the production ID is different for any production. A production (also known as transformation in the dirac speak) is a transformation of an input file into an output file.
So the first production is the generation of the input files (prodID 1335), then the simulation of the generator file (production 1336). and so on

Or use dirac-ilc-get-info -p 1338
which tells you the input files are from Prod 1337 and go down the chain.

2) If you run anajob on the slcio file, it will contain the mokka steering file at the top of the output.

3) If there are no B-mesons/baryons decays in the generator file it is probably not needed.
But it is probably better to provide it.

Subject: Re: Re-running a given simulations set
Posted by protopop on Mon, 03 Feb 2014 15:38:54 GMT
View Forum Message <> Reply to Message

Thanks a lot for the reply. That clarifies a lot of issues.

In view of the included Mokka steering file (quoted in red in my previous post), any idea how to find its source and correct the (runtime) error below ?

-------- WWWW ------- G4Exception-START -------- WWWW -------
*** G4Exception : GeomNav1002
issued by : G4Navigator::ComputeStep()
Track stuck or not moving.
Track stuck, not moving for 10 steps
in volume -Slab- at point (48.8322,-1897.95,772.416)
direction: (0.352256,0.813033,-0.463566).
Potential geometry or navigation problem !
Trying pushing it of 1e-07 mm ...Potential overlap in geometry!

*** This is just a warning message. ***

-------- WWWW -------- G4Exception-END --------- WWWW -------

Even if it says it's jus a warning, and we're only talking about 10^{-7}mm, the "Potential overlap in geometry!" bit worries me.

Subject: Re: Re-running a given simulations set
Posted by sailer on Mon, 03 Feb 2014 15:54:45 GMT
View Forum Message <> Reply to Message

To check for overlaps:
In the geant4 shell (or in a macro)
run
/geometry/test/recursive_test
Have a look at the built in documentation (run "help" in geant4 shell) for more information and other tests.
Note that false positives are possible.

You can also do a visual inspection of the geometry.
Or check the parameters printed for the different drivers (e.g., ECal position and size vs. hcal position)

Subject: Recalibration needed after change in geometry ?
Posted by protopop on Mon, 24 Feb 2014 16:42:46 GMT
View Forum Message <> Reply to Message

Let's say I tweak the ECAL geometry and I run a bunch of Mokka simulations with the new geometry.

When I want to run reconstruction afterwards, do I need to do any recalibration of the reconstruction parameters (for Pandora ?) or it's all done automatically based on the new Gear file which I provide?

Subject: Re: Recalibration needed after change in geometry ?
Posted by gaede on Fri, 28 Feb 2014 12:15:21 GMT
View Forum Message <> Reply to Message

It depends very much on the changes that you make to the geometry.

For example changing the thickness of the scintillator clearly requires new calibration, as would a change of the cell sizes.
In case of doubt, it is best to contact the Pandora experts directly.
-Frank.
