McCad user guide

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1. Overview

McCad is a Monte Carlo (MC) CAD-based modeling program. It can be used for converting CAD model to half-space Constructive Solid Geometry (CSG), tessellated (faceted) solid and unstructured mesh for MC codes. Currently supported MC codes are MCNP5/6 (CSG and mesh geometry), TRIPOLI-4 and Geant4 (half-space CSG and tessellated solid). McCad has been integrated in the open-source SALOME platform, and relies on user interfacing. This manual is intended to provide a guide on how to generate an MC codes input file using McCad.

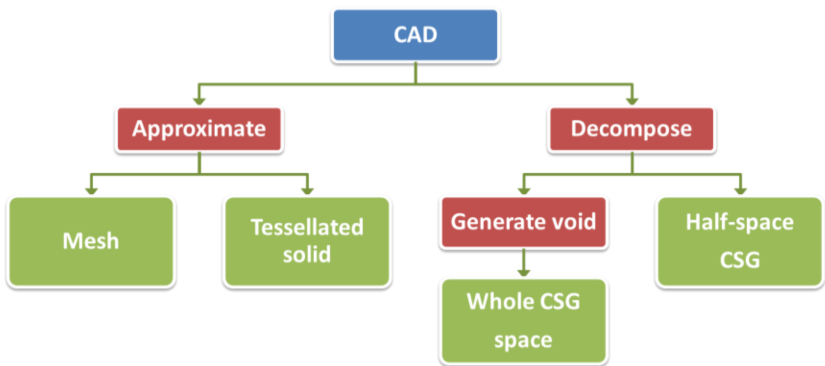


Fig 1.1 McCad workflow

1.1 General workflow

The general workflow of McCad is shown in Fig 1.1. The McCad conversion processes include:

- Decomposing process: CAD model are decomposed into numbers of solids which can be described as so-called *sign-constant solids* (or half-space solid). These solids are formed by intersections of Boolean half-spaces. Note: not all the solid are able to be decomposed by McCad, user have to check using the "Check before convert" function and try to decompose those solid by themselves, and use "Mark as decomposed" to skip the automatic decomposition. See the following chapters for details.
- Generating void: In MC codes such as MCNP, the whole space should be unambiguously defined. McCad provides a conversion process to describe the void space as well as the solid geometries. The solids has to be sign-constant solids.
- Approximating process: CAD geometry can be approximate by tessellated solid or unstructured mesh. Tessellated solid describes the geometry using triangle or quadrangle facets. The unstructured mesh discretizes the CAD geometry using mesh elements. McCad uses open-source meshing tools *Tetgen* and *Netgen* to generate the unstructured mesh, and use OpenCASCADE library to generate the tessellation facets.

For generating input file for specific MC, it needs to carry out following processes:

- **MCNP traditional CSG: Decomposition -> Void Generation**. This conversion is performed automatically, unless decomposition failed for some parts. Note: The "Generate void" should be activated in the Preference page.
- **MCNP6 hybrid geometry: Decomposition + Approximiation -> Void generation**. Some part of the geometry can be model with unstructured mesh, others are remain to be CSG. The detail procedure can be found in the tutorial. Note: the
- **TRIPOLI CSG:** similar as MCNP traditional CSG.
- **Geant4 hybrid geometry: Decomposition + Approximiation**. Geant4 does not require void generation. Some part of geometry which are decomposed is converted into CSG, and those not being decomposed (or cannot be decomposed) can be converted to tessellated solid. Note: To generate the tessellated solid, the "Force decompose" muss deactivated in the Preference page.

1.2 Units

The McCad is using ***mm*** as unit, and the model is converted internally into ***cm*** for MC codes. For unstructured meshes, there is no specific requirement on unit, but the *Length Conversion Factor* should be set properly in preference page.

2. Compilation and installation

Note: You don't need to compile the code if you are only using it. You can find the section 2.2 for how to download and install the binary executables. Compilation is needed only if you change the code.

2.1 compilation

2.1.1 Compilation and installation on Linux system

Step 1. Installing Salome platform

- Download Salome_7.4.0 platform from http://www.salome-platform.org/downloads/previous-versions/salome-v7.4.0 .
    - Under the list "Binaries for officially supported Linux platforms", Choose the version which is closest to your OS.
    - You need to register for downloading Salome.
- Extract the zip file into a folder, then go into this folder and run the script "runInstall". A install wizard window will come out.
- Click always "Next", and keep the default value if you don't care.
    - If you like, you can change the installation folder;

- It is highly recommended to install all modules except DOCUMENTS, in order to avoid unncessary error.
- There might be warnings on "cppunit" libraries and so on, it won't affect the use of Salome.

- We abbreviate the Salome install folder as $SALOME

Step 2. Compiling McCad

- Download this package, unzip and rename it as "MCCAD_SRC_0.5.0", place it under $SALOME.
- We need to add MCCAD package into the Salome environment for compiling it. In the following changes, back-up those file before changing them.
    - Open $SALOME/build.sh (or $SALOME/build.sh), find line 32, and add "MCCAD" into the list. At the end it looks like: def_modules="${def_modules} RANDOMIZER SIERPINSKY ATOMIC ATOMGEN ATOMSOLV DOCUMENTATION MCCAD"
    - Add the following environment variables to the end of $SALOME/env_build.sh (if you are using csh, then $SALOME/env_build.csh)

```
  #------ MCCAD ------
  export MCCAD_ROOT_DIR=${INST_ROOT}/MCCAD_0.5.0
  if [ -n "${ENV_FOR_LAUNCH}" ] ; then
    if [ "${ENV_FOR_LAUNCH}" = "1" ] ; then
      exportp PATH ${MCCAD_ROOT_DIR}/bin/salome
      exportp LD_LIBRARY_PATH ${MCCAD_ROOT_DIR}/lib/salome
      exportp PYTHONPATH
${MCCAD_ROOT_DIR}/bin/salome:${MCCAD_ROOT_DIR}/lib/python${PYTHON_VERSION}/site-
packages/salome
    fi
  fi
  ##
  #------ MCCAD_src ------
  export MCCAD_SRC_DIR=${INST_ROOT}/MCCAD_SRC_0.5.0
```

- Go to $SALOME, run the following command to compile McCad:

./build.sh MCCAD

- After the compilation, You can find the binaries in $SALOME/INSTALL/MCCAD_0.5.0. Copy this folder "MCCAD_0.5.0" to $SALOME folder.

Step 3. Installing and running McCad

- Be sure that your $SALOME/MCCAD_0.5.0 folder have following folders:
    - bin
    - lib
    - share
    - adm_local
    - idl
    - include

- Open $SALOME/KERNEL_7.4.0/salome.sh, add the following environment variables into this file.

```
#------ MCCAD ------
export MCCAD_ROOT_DIR=${INST_ROOT}/MCCAD_0.5.0
if [ -n "${ENV_FOR_LAUNCH}" ] ; then
  if [ "${ENV_FOR_LAUNCH}" = "1" ] ; then
    exportp PATH ${MCCAD_ROOT_DIR}/bin/salome
    exportp LD_LIBRARY_PATH ${MCCAD_ROOT_DIR}/lib/salome
    exportp PYTHONPATH
${MCCAD_ROOT_DIR}/bin/salome:${MCCAD_ROOT_DIR}/lib/python${PYTHON_VERSION}/site-
packages/salome
  fi
fi
##
#------ MCCAD_src ------
export MCCAD_SRC_DIR=${INST_ROOT}/MCCAD_SRC_0.5.0
```

- In your desktop, create a new file "runSalome.sh" and put following text into this file(replacing $SALOME with actual path!!):

```
#!/bin/bash
source $SALOME/KERNEL_7.4.0/salome.sh
$SALOME/salome_appli_7.4.0/salome --module=GEOM,SMESH,PARAVIS,MCCAD
```

- Under Desktop, make this file as executable script using this command:

chmod +x ./runSalome.sh

- You can run McCad-Salome with running this script now.

2.1.2 Compilation and installation on Windows system

ATTENTION: In this following step you need to have Visual Studio 2010. Without it or with other version of Visual Studio will failed.

Step 1. Installing Salome platform

- Download Salome_7.4.0 platform from http://www.salome-platform.org/downloads/previous-versions/salome-v7.4.0 .
    - The version you need is "SALOME SDK self-extracting archive for 64bits Windows".
    - You need to register for downloading Salome.
- Extract the Package into a folder, here brief as $SALOME (be sure to replace it with actual path during installation).
- Open cmd.exe from Windows Start menu, change folder to $SALOME\WORK, and execute "compile.bat". Please take a coffee, because it takes hours. When the compilation finished, Salome platform will be ready.

Step 2. Compiling McCad

- Download the McCad-Salome package, unzip it.
- In $SALOME\MODULES, Create a folder "MCCAD", put the source code inside, and rename the source code folder as "MCCAD_SRC"
- We need to add McCad into the Salome environment. First make a backup of file $SALOME\WORK\set_env.bat
    - in line 55, add "MCCAD" into the "list" (add it anywhere inside the parenthesis, separate with at least one whitespace)
    - in line 81, add "GUI GEOM MED SMESH PARAVIS MCCAD" into "env_m_list"(add them inside the parenthesis, separate with at least one whitespace).
- In file $SALOME\WORK\compile.py, add the following line to line 35

all_modules.append('MCCAD')

- Open cmd.exe from Windows Start menu, change folder to $SALOME\WORK, and execute command:

compile.bat MCCAD

- The compilation will finished in a few minutes.

Step 3. Installing and using McCad

- To run McCad, start cmd.exe in the Windows Start menu, and run the following command:
    - $SALOME\WORK\run_salome.bat --module=MCCAD
    - If you want to start also geometry, meshing and visualization module, using command: $SALOME\WORK\run_salome.bat --module=GEOM,SMESH,PARAVIS,MCCAD
- One more easy way to run the program is:
    - right-click $SALOME\WORK\run_salome.bat and "Send to -> Desktop (short-cut)";
    - right-click the short-cut link in the Desktop, choose "properties";
    - Behind the value of "Target", add " --module=MCCAD" or "--module=GEOM,SMESH,PARAVIS,MCCAD"(with a whitespace in the front). Click "OK". Next time you can start McCad with this short-cut link.

2.2 Installation

This section is for installing the precompiled binary executable. If you have done the compilation in Section 2.2, you don't need to go through this section.

2.2.1 *Installation and run McCad on Windows*

- Download Salome_7.4.0 platform from http://www.salome-platform.org/downloads/previous-versions/salome-v7.4.0 .
    - The version you need is "binaries self-extracting archive for 64bits Windows".
    - You need to register for downloading Salome.
- Extract the Package into a folder, here brief as $SALOME (be sure to replace it with actual path during installation).
- Download the "MCCAD_0.5.1-beta_for_Salome_7.4.0_WIN64.zip", extract it and place it under $SALOME/MODULES. Under $SALOME\MODULES\MCCAD there are three folder:
    - bin
    - lib
    - share
- We need to add MCCAD into the Salome environment. First make a backup of file $SALOME\env_launch.bat
    - in line 29, add "MCCAD" into the "m_list" (add it anywhere inside the parenthesis, separate with at least one whitespace)
    - in line 42, add "GUI GEOM MED SMESH PARAVIS MCCAD" into "env_m_list"(add them inside the parenthesis, separate with at least one whitespace).
- To run McCad, start cmd.exe in the Windows Start menu, and run the following command:
    - $SALOME\run_salome.bat --module=MCCAD
    - If you want to start also geometry, meshing and visualization module, using command: $SALOME\run_salome.bat --module=GEOM,SMESH,PARAVIS,MCCAD
- One more easy way to run the program is:
    - right-click $SALOME\run_salome.bat and "Send to -> Desktop (short-cut)";
    - right-click the short-cut link in the Desktop, choose "properties";
    - Behind the value of "Target", add " --module=MCCAD" or " --module=GEOM,SMESH,PARAVIS,MCCAD"(with a whitespace in the front). click "OK". Next time you can start McCad with this short-cut link.

2.2.2 *Installation and run McCad on Linux*

- Download Salome_7.4.0 platform from http://www.salome-platform.org/downloads/previous-versions/salome-v7.4.0 .
    - Under the list "Binaries for officially supported Linux platforms", Choose the version which is closest to your OS.
    - You need to register for downloading Salome.

- Extract the zip file into a folder, then go into this folder and run the script "runInstall". A install wizard window will come out.
- Click always "Next", and keep the default value if you don't care.
    - If you like, you can change the installation folder;
    - It is highly recommended to install all modules except DOCUMENTS, in order to avoid unncessary error.
    - There might be warnings on "cppunit" libraries and so on, it won't affect the use of Salome.
- We abbreviate the Salome install folder as $SALOME.
- Download the corresponding McCad binary. Try to choose the version closest to your OS. If it does not work after the following installation, the only solution is to download the McCad source code and compiled the binary by yourself.
- Extract the zip file and place it under $SALOME. Be sure that your $SALOME/MCCAD_0.5.1 folder have following folders:
    - bin
    - lib
    - share
    - adm_local
    - idl
    - include
- Open $SALOME/KERNEL_7.4.0/salome.sh, add the following environment variables into this file.

```
#------ MCCAD ------
export MCCAD_ROOT_DIR=${INST_ROOT}/MCCAD_0.5.1
if [ -n "${ENV_FOR_LAUNCH}" ] ; then
  if [ "${ENV_FOR_LAUNCH}" = "1" ] ; then
    exportp PATH ${MCCAD_ROOT_DIR}/bin/salome
    exportp LD_LIBRARY_PATH ${MCCAD_ROOT_DIR}/lib/salome
    exportp PYTHONPATH
${MCCAD_ROOT_DIR}/bin/salome:${MCCAD_ROOT_DIR}/lib/python${PYTHON_VERSION}/site-
packages/salome
  fi
fi
##
#------ MCCAD_src ------
export MCCAD_SRC_DIR=${INST_ROOT}/MCCAD_SRC_0.5.1
```

- In your desktop, create a new file "runSalome.sh" and put following text into this file(replacing $SALOME with actual path!!):

```
#!/bin/bash
source $SALOME/KERNEL_7.4.0/salome.sh
$SALOME/salome_appli_7.4.0/salome --module=GEOM,SMESH,PARAVIS,MCCAD
```

- Under Desktop, make this file as executable script using this command:

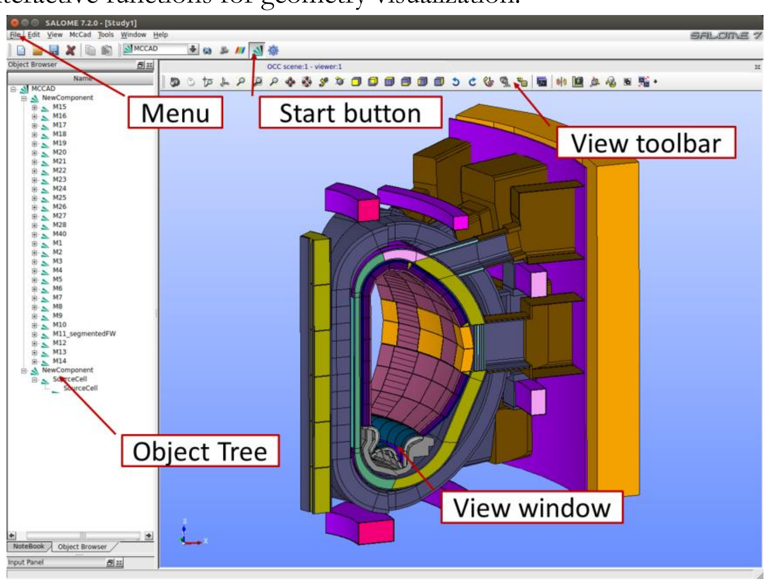
chmod +x ./runSalome.sh

- You can run McCad-Salome with running this script now.

3. User manual

Note: Before reading this Chapter, you are suggested to go through first a tutorial in the Chapter 4 targeting your interested codes. When you need to know further the details, you can look up the corresponding item for explanation.

Fig 3.1 shows the GUI of McCad program. It is consisted of five blocks:

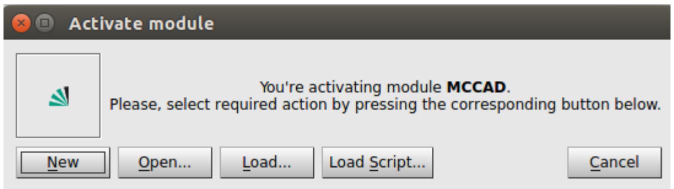
- **Start buttons**: buttons to start a module in SALOME platform.
- **Menu**: Functions implemented in the program. The *File* and *McCad* menu have implementations functions for McCad.
- **Object tree**: geometry objects.
- **View window**: geometry displaying window. and
- **View toolbar**: interactive functions for geometry visualization.
- 

- Fig 3.1 McCad GUI

3.1 starting McCad

For starting McCad, we need to start SALOME platform first, and then choose the McCad module:

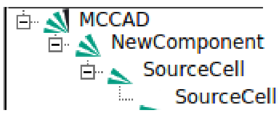
- Start SALOME platform, click the McCad start button: 
- then the following dialog pop-up:



- Click *New* for create a new McCad project.
- Then the McCad GUI will show as Fig. 2.1.

3.2 Object tree

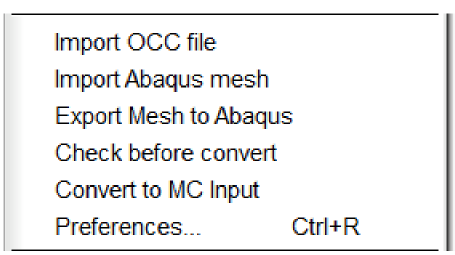
The object tree are organized as following :



- **MCCAD**: The Root object. Every module has a unique root object. It is the father object of all objects in this model.
- **Component**: The component organizes the model in an assembly-like structure. It contains groups of geometry parts. When generated unstructured mesh for all the geometry in this component, the component should be assigned with a CAD solid as envelop for the meshes.
- **Group**: The group organizes the geometries with the same material. Material information is assigned for these groups.
- **Part**: A part is an end node of the object tree. It represents a geometry entity that has only one material composition. It might contain just one CAD solid, or several CAD solid together as a compound, or CAD solids with an unstructured mesh.

3.3 File menu

In the file menu, the following functions have been implemented:



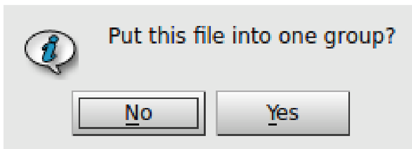
- *Import CAD file*: importing Open CASCADE CAD files, including BRep, STEP, IGES format.
- Import Abaqus mesh: importing mesh files in Abaqus format. Noted that only this interface does not support general Abaqus format, but specific format similar as the file exported from McCad.
- *Export Mesh to Abaqus*: export meshes to Abaqus mesh format. Mesh files in this format can be used for MCNP6 unstructured mesh geometry.
- *Check before convert*: Check the model before been converted into the MC input file. The program will conduct necessary check on the validity of the model, and all the message will be shown in the

command window where SALOME is started. The messages includes geometry decomposition, material assignment, volume information, etc.

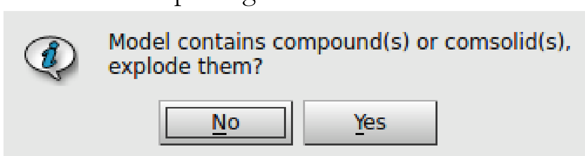
- *Convert to MC input*: converted the model into an input file of a chosen MC code. The target MC code can be chosen from the *Preferences* . Before conversion, an automatic check on the model will be performed.
- *Preferences*: see section in below.

3.3.1 Import CAD file

For importing an OCC file, the following dialog helps users to organizing the structure:



- If more than one file are imported, this dialog will pop-up and ask if put these files into one group.
- It is better not putting CAD solids in different file into one group, order to distinguish the data.



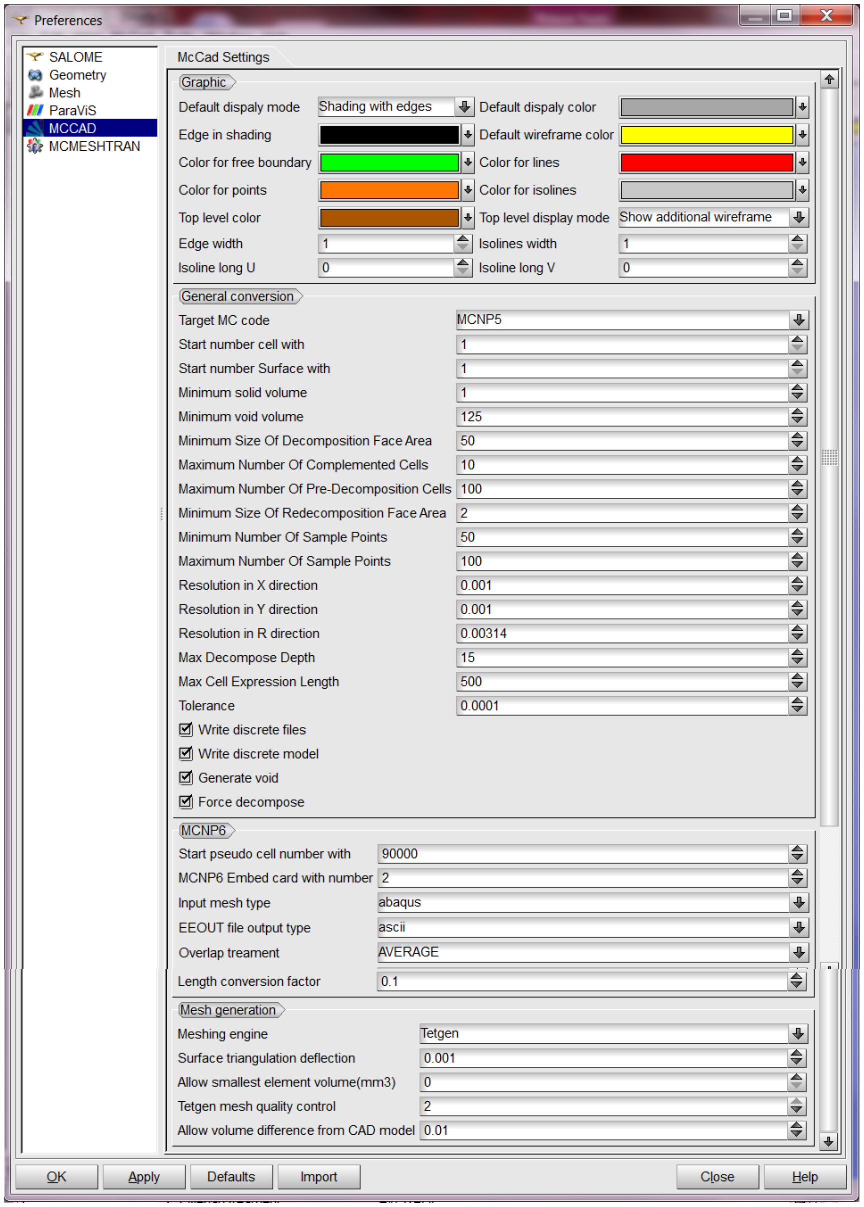
- If a file contains more than one CAD solid, then it will ask if explode it.
- User can explode a compound solid or fuse some solid as a compound later in the GUI, if the decision is changed later.
- For generating mesh using McCad's own meshing function, it requires exploding the compound because only one mesh can be assigned for one geometry part.

For ease of use, user can append the imported geometry to exiting components or groups by selecting objects.

- If multiple files are imported
    - when any component/group/part selected, the geometry will be append in current component as a new group
- If only one file is imported
    - when a group/part is selected, the geometry will be append at the end this the current group
    - when a component is selected, the geometry will be append to the component as a new group
- In other case, the imported data will be put in a new component.

3.3.2 Preference page

The preference page manages all the parameters used for the conversion. Currently it is not well organized and well implemented. However, only few of them should be take care before conversion.



- Graphic
  - *Default display mode* - allows to choose between wireframe, shading, or shading with edges.
  - *Default shading color* - allows to select default shading color.
  - *Edges in shading* - allows to select default edges color in shading mode.
  - *Default wireframe color* - allows to select default wireframe color (to be applied to any lines not being free boundaries or isolated lines).
  - *Color of free boundaries* - allows to select default color for free boundaries.
  - *Color of edges, vectors and wires* - allows to select default color for edges, vectors and wires (isolated lines).
  - *Color of points* - allows to select default color for vertices.
  - *Color of isolines* - allows to select default color for isolines.

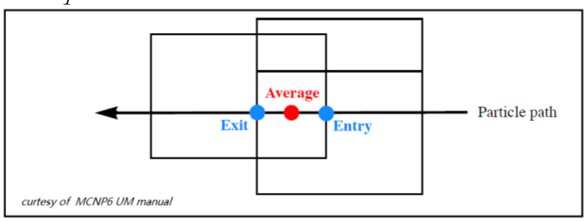
- *Top level color* - allows to select default color for objects which were brought to the viewer foreground.
- *Top level display mode* - allows to select default top level display mode between:
  - *Show additional wireframe actor* - allows to have the shading actor at its usual place (in the back) and add the additional wireframe actor in the viewer foreground.
  - *Keep current display mode* - allows to use current display mode of object.
  - *Wireframe* - allows to switch display mode to wireframe mode after "top-level" operation.
  - *Shading* - allows to switch display mode to shading mode after "top-level" operation.
  - *Shading With Edges* - allows to switch display mode to shading with edges mode after "top-level" operation.
- *Edges width* - allows to define default width of the edges.
- *Isolines width* - allows to define default width of the isolines.

- General conversion
  - ***Target MC code***
    - *MCNP5*: converting CSG for MCNP5
    - *MCNP6*: converting CSG or unstructured mesh for MCNP6
    - *TRIPOLI-4*: converting CSG for TRIPOLI-4
    - *GDML*: converting CSG and tessellated solid for Geant4

  - *Start number cell with*: Initiate cell number in output
  - *Start number surface with*: initiate surface number in output
  - *Minimum solid volume*: all volume smaller than this will be neglected
  - *Minimum void volume with*: make sure the void volume don't become too small
  - *Minimum size of decomposition face area*: first cut surface area
  - *Maximum number of complemented cells*: how many complement operator can be used per void volume
  - *Maximum number of pre-decomposition cells*: Set the upper bound for number of void volumes volume after first cut
  - *Minimum size of redecomposition face area*: recut if maximum number of cells is exceed
  - *Minimum number of sample points*: lower bound for discretization along a face's edge
  - *Maximum number of sample points*: upper bound for discretization along a face's edge
  - *Resolution in X direction*: resolution of discretization in X direction
  - *Resolution in Y direction*: resolution of discretization in Y direction
  - *Resolution in R direction*: resolution of discretization in R direction
  - *Max decompose depth*: not applicable
  - *Max cell expression length*: not applicable
  - *Tolerance*: tolerance used in decomposition
  - *Write discrete files*: Write *.CollisionFile of input solids (yes/no)
  - *Write discrete model*: Write *.voxel files for input solids (yes/no)
  - ***Generate void***: conduct void generation or not (yes/no)
  - ***Force decompose***: automatic decompose parts which is not decomposed (yes/no). CAUTION: set "no" only if converting MCNP6/GDML file. When converting MCNP6 model, the parts which are not decomposed will checked if they have meshes. When

converting to GDML, the parts which is not decomposed will be converted to tessellated solids. For target code of MCNP5/TRIPOLI, uncheck this might cause error.

- MCNP6
  - *Start pseudo cell number with*: the initiate cell number of pseudo cell
  - *MCNP6 Embed card with number*: the initiate embed card number in the input file
  - *Input mesh type*: type of mesh for mcnp6, currently abaqus only
  - *EEOUT file output type*: MCNP6 unstructured mesh output format
    - ASCII: text format. Enable to be processed by McMeshTran
    - Binary: unformatted
  - *Overlap treatment*: treatment in MCNP6 for the overlapping

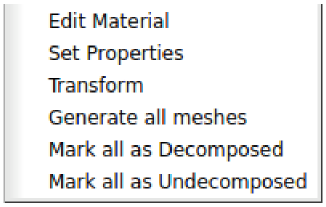


    - Exit: take the exit point as the intersection point
    - Entry: take the entry point as the intersection point
    - Average: take the middle point as the intersection point
  - ***Length conversion factor***: factor to convert unit used for the mesh to centimeter. usually mesh are in meter or millimeter, thus the value usually be 100 or 0.1
- Mesh generation
  - ***Mesh engine***: meshing tool to be chosen
    - Tetgen: for generating TT mesh. Mesh are very coarse for use in MC geometry
    - Netgen: generating Finite element mesh. Mesh are also coarse and not enable to adjust the size
  - *Surface triangulation deflection*: set the relative tolerance in faceting the solid, the value is adjust according to the size of each solid. The faceting accuracy will change as well as the mesh volume deviation.
  - *Allow smallest element volume (mm^3)*: remove those mesh elements with very small volume sizes.
  - *Tetgen mesh quanlity control:* Not applicable.
  - *Allow volume difference from CAD model*: Allowable difference between CAD volume and mesh volume. This value set limits for checking if the meshes are consistent with the CAD solids.

Most of the options and parameters can be used in default. Only be careful on the option and parameters in ***Italic Bold***.

3.4 McCad menu

In the McCad menu, the following functions have been implemented:

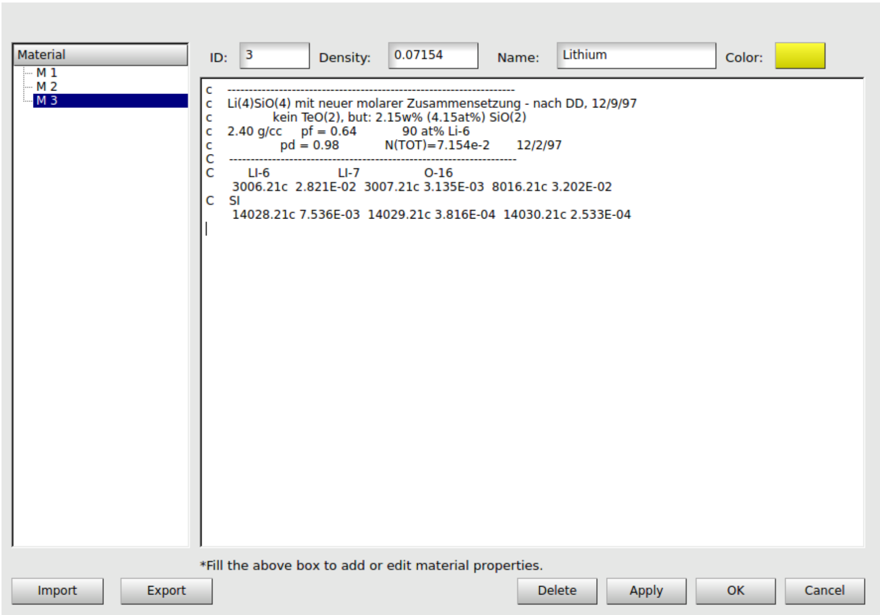


- *Edit Material*: open the material editing window. See Section below for detail.
- *Set properties*: set the Material, color, importance, etc. properties for geometry objects. See Section below for detail
- *Transform*: Transform the selected geometry objects.
- *Generated all meshes*: generated Tesellation-Tetrahedralization (TT) mesh for all the geometries. For generating meshes for some part of the geometry, use *Generate Tetrahedral mesh* in the pop-up menu instead.
- *Mark all as Decomposed*: Marking the geometry as already being decomposed.

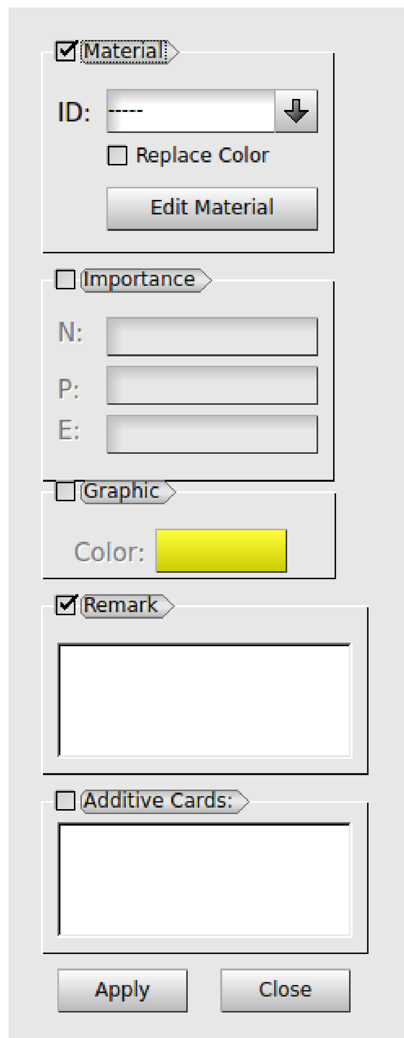
3.4.1 Edit material

The material management window is shown as following. Currently the material management is tested only for MCNP and TRIPOLI code. For GDML, the material name and density are exported, but the information in the text box is not usable yet.

- *Material tree*: for displaying the material .
    - The displaying name of the material is: “M”+ID
- *ID*: material identification number which should be unique. This is initially design for MCNP.
- *Density*: Atom density (positive value) or mass density (negative value)
- *Name*: Name for the material. Highly recommend to be unique, because it will be used for code like Geant4 for material identification.
- *Color*: Color of this material. User can choose to change color of geometry parts according to this color.
- *(Text box):* For filling material composition.
- *Import*: importing the material file. The material a XML file is the file exported by this window.
- *Export*: exporting the material into a XML file, which can be import for the next time.
- *Delete*: delete the selecting material



- *Apply*: confirm the current input data
- *OK/ Cancel*: confirm and exit/ just exit



3.4.2 Set properties

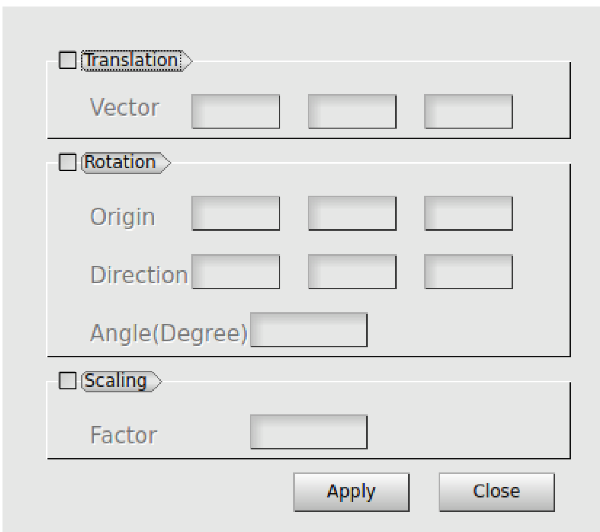
The properties of component/group/part can be set using the Properties dialog. This dialog is interactive with the object tree, and enable to assigned properties for multiple objects. The properties assigned for object in upper level will overwrite that of objects in lower level. The properties will be gray out if not available for the current selected object.

- *Material properties*
  - Only available when groups are selected.
  - *ID*: for choosing the material, the material is shown with ID + name
  - *Replace color*: if checked, the color for all the parts in this group will be overwrite with the material color
  - *Edit material*: show the material management window.
- *Importance* (Only for MCNP)
  - *MCNP* importance for cells, used for variance reduction.
  - *N*: neutron importance
  - *P*: photon importance
  - *E*: electron important (NOT APPLICABLE)
- *Graphic*
  - *Color*: for setting the color

- *Remark* (Test MCNP only)
    - Adding remarks for this object, useful for marking some geometry parts for tallying…
    - Independent remark for component/group/part, not overwriting.
- *Additive Cards* (Only for MCNP)
    - Add additional card for specific parts, e.g. fill card, universe card, etc.
- *Apply/Close*: confirm or exit.

3.4.3 Transformation

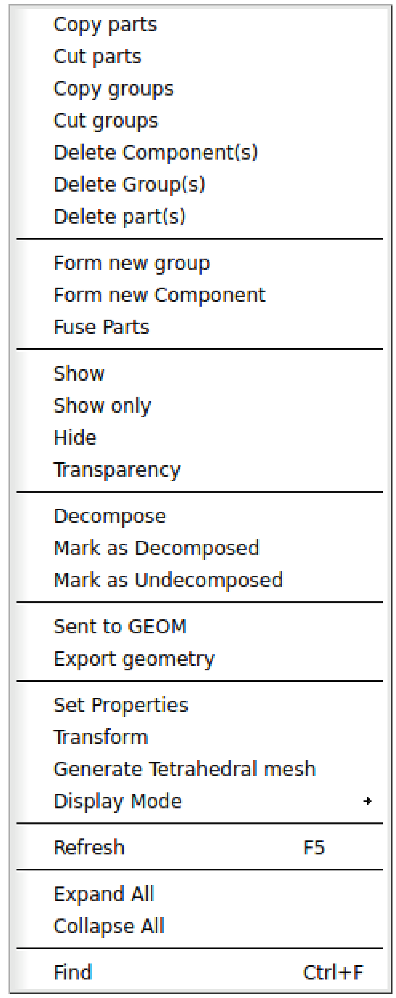
Transformation functions are provided for manipulating the model. They are activated when the box are checked.



- *Translation*
    - Move the model according to the vector
    - *Vector*: X, Y, Z value of the vector
- *Rotation*
    - rotate the model according to the Axis and angle
    - *Origin*: origin point of the axis
    - *Direction*: direction vector of the axis
    - *angle*: rotate angle in degree
- *Scaling*
    - Scaling the model referring to the base point (0,0,0). Used frequently for unit conversion.
    - *Factor*: scaling factor
- If more than one transformations are checked, the operation order will be Translation->Rotation->Scaling.

3.5 Pop-up menu

Pop-up menu is shown with right-click on the objects. The menu is different when different objects are selected.



- *Copy parts/groups*: copy the parts/groups
  - If the clipboard has data which are not clear, a warning will be given.
- *Cut parts/groups*: cut the parts/groups
- *Paste parts/groups*: paste the copied/cut parts/groups in the selected location.
  - Based on what kind is selected, the object will be append to the end of the selected object.
- *Delete components/groups/parts*: remove them.
- *Form new components/groups*: if multiple parts are selected, they can form a new group. Similarly for groups.
- *Fuse parts*: fuse the part as a compound.
- *explode parts*: if the part is a compound, explode to solids.
- *Show*: display the geometry
- *Show only*: show only selected geometry and hide others
- *Hide*: erase the geometry from the view window
- *Hide All:* Hide all object. NOT APPLICABLE
- *Transparency*: set the Transparency of the solid. Currently working only on Shading mode
- *Display mode*:
  - Wireframe: only display the edges
  - Shading: show the faces
  - Shading with wireframe: show the edges with faces
- *Decompose*: cut the solid into sign-constant solid
- *Recover*: undo the decomposition and recover the geometry
- *Mark as decomposed*: When you are sure the selected solids are already decomposed or do not need to be decomposed, you can mark it as decomposed to save computation time. Make sure that the decomposition is not needed and USE IT WITH CAUTION!
- *Mark as undecomposed*: mark the solid as undecomposed to force decomposing
- *Send to GEOM*: send the selected geometry to SALOME GEOM module
- *Send to SMESH*: send the meshes to SALOME SMESH module. Only effective when these parts have mesh data
- *Export geometry*: exporting the geometry to BRep, STEP or IGES format.
- *Set properties*: Set properties for the selected objects, see Section 3.4.2.
- *Transform*: make transformation. See Section 3.4.3.
- *Generate Tetrahedral mesh*: Generate TT mesh on the selected objects. ATTENTION: the part should be exploded in advance if it is a compound; the old mesh will be overwritten.
- *Clear mesh*: clear the mesh data
- *Refresh*: refresh the tree;

- *Expand All*: show all the children objects under this object
- *Collapse All*: collapse the selected object
- *Find*: Search for an object with given string
- *Dump View*: capture the screen and save as an image

3.6 View toolbar

These tools are provide by SALOME GUI viewer, details see http://docs.salome-platform.org/latest/gui/GUI/occ_3d_viewer_page.html. They are listed for convenience.

- *Dump View* - exports an object from the viewer in bmp, png or jpeg image format
- *Interaction style switch* - allows to switch between standard and "keyboard free" interaction styles.
- *Zooming style switch* - allows to switch between standard (zooming at the center of the view) and advanced (zooming at the current cursor position) zooming styles.
- *Show/Hide Trihedron* - shows or hides coordinate axes
- *Fit all* - allows to select a point to be the center of a scene representing all displayed objects in the visible area
- *Fit area* - resizes the view to place in the visible area only the contents of a frame drawn with pressed left mouse button.
- *Zoom* - allows to zoom in and out. The short-cut is “ctrl+left mouse button”
- *Panning* - if the represented objects are greater that the visible area and you don't wish to use *Fit all* functionality, click on this button and you'll be able to drag the scene to see its remote parts. The short-cut is “ctrl+middle mouse button”
- *Global panning* - represents all displayed objects in the visible area.
- *Change rotation point* - allows to to choose the point around which the rotation is performed
- **Rotation** - allows to rotate the selected object using the mouse. The short-cut is “ctrl+right mouse button”
- These buttons orientate the scene strictly about coordinate axes: *Front*, *Back*, *Top*, *Bottom*, *Left or Right* side.
- *Rotate counterclockwise* - rotates view 90 ° counterclockwise.
- *Rotate clockwise* - rotates view 90 ° clockwise.
- *Reset* - restores the default position (isometric) of objects in the scene.
- *Memorise view* - saves the current position of objects in the scene
- *Restore view* - restores the saved position of objects in the scene
- *Clone view* - opens a new duplicate scene.
- *Clipping* allows creating cross-section views (clipping planes) of your mesh.
- *Scaling* - represents objects deformed (stretched or stuffed) along the axes of coordinates.
- *Graduated axes* - allows to define axes parameters and graduate them

- *Toggle ambient* **light** - toggle "keep only ambient light" flag on/off.
- *Minimize*/*Maximize* - these buttons allow switching the current view area to the minimized / maximized state.
- *Synchronize view* - allows to synchronize 3d view parameters.

4. Tutorials

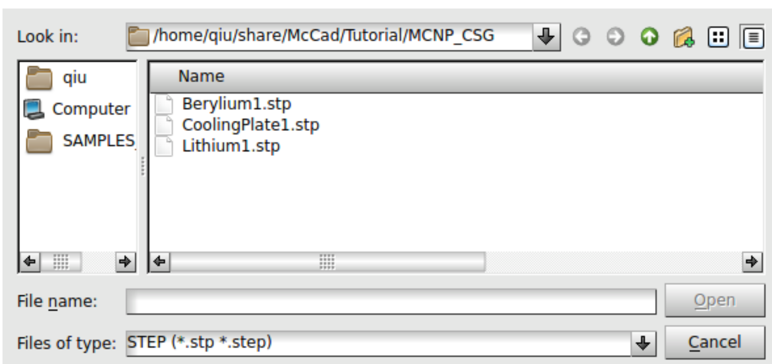
These tutorials provide example for showing how to create a input file for MC simulation. It need to be noticed that the input file create by McCad is not completed, therefore user should add necessary parameters before using it for calculations.

4.1 MCNP/TRIPOLI traditional CSG conversion

This tutorial demonstrates how to generate a traditional MCNP input file. In McCad the target MC code option is "MCNP5/TRIPOLI".

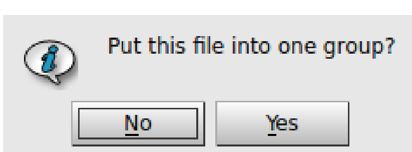
Step 1. Importing the geometry.

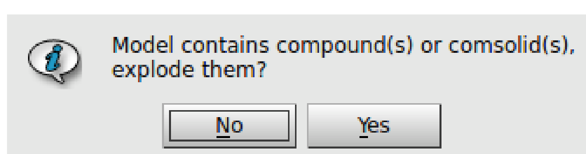
- Open menu *File->Import CAD file*, select files type to be STEP (default), and select all the three STEP files and open: Berylium1.stp, CoolingPlate1.stp and Lithium1.stp.



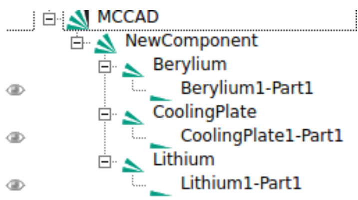
- A dialog is shown and ask whether put CAD solid in these file into one group, we choose *No*.
- Then it will ask if explode the compounds. We choose *Yes* to explode them.

*NOTE: Sometimes only one solid is shown after explode, it might because the compound contains free faces or edges which have been filtered, or the compound just contain one solid.



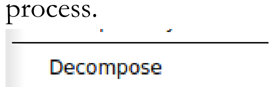


- At then end the object tree looks like following. We can right-click and show the geometry.
-

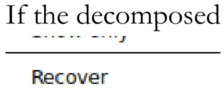


Step 2. Decompose the solids (optional).

- Right-click the "NewComponent", and click *Decompose*, all the parts in this component will be decomposed. If this step is skipped, all the parts will still be decomposed during the conversion process.

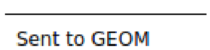


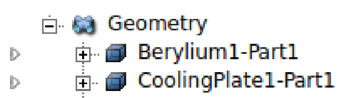
- If the decomposed solid is too complex, we can *Recover* the solid and do manual cut on it first.



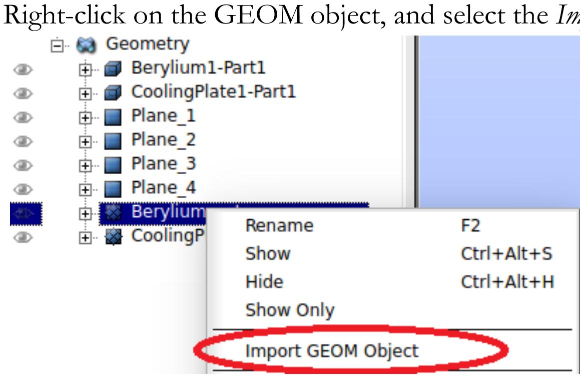
Step 3. Manual cut the solid (optional).

- Select the Berylium-Part1 and Lithium-Part1, right-click and choose *Send to GEOM*.
- Then open the GEOM module by clicking on the GEOM start button (as highlighted). We can find the sent geometries are under the object tree of GEOM.

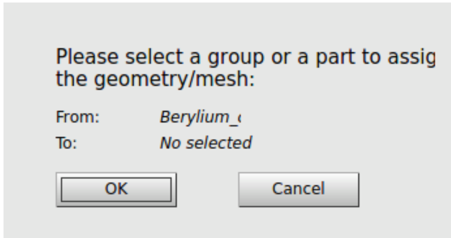




- Then cut the solid using GEOM functionalities. The idea is create a plane and use it for partitioning the solid. For detail instruction see SALOME user guide. http://docs.salome-platform.org/latest/gui/GEOM/partition_page.html.
- After the cutting, the data can be imported and assigned for existing objects.
  - Return to McCad by click the *McCad start button*.
  - Right-click on the GEOM object, and select the *Import GEOM Object* as show as follow.



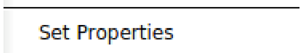
  - A dialog will come out and ask to assign to which object. choose the Berylium-Part1 in the McCad object tree because we want to update it with cut model. The dialog will update with the object name. Click OK.
- Note: For further information on interactive decomposition please read the Section 4.4.
-



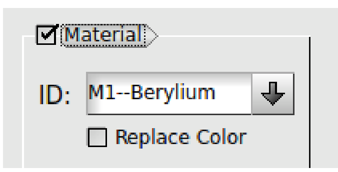
- Visualize the object in order to confirm the correctness.

Step 4. Assign material.

- Open material management window from menu McCad->Edit Material, click *Import* to and choose material.xml file.
- Modify the data if needed. Then Click *OK* or *Cancel* to exit.
- Right-click on the group Berylium, and choose Set properties to show the *Properties* window



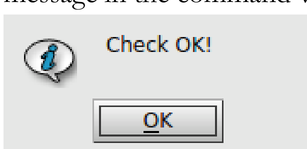
- Set the material to be M1--Berylium, and select a color for this group, and then click Apply.

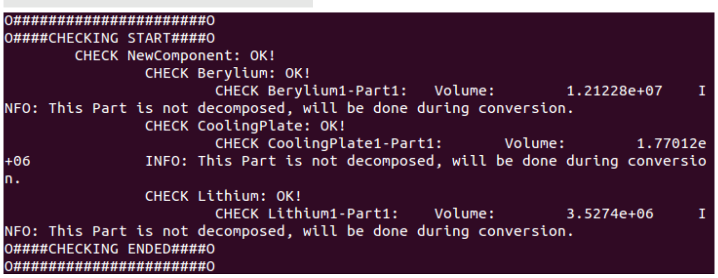


- Similarly for CoolingPlate and Lithium.

Step 5. Conversion

- First check if the *MCNP5* or *TRIPOLI* option is selected in Target MC code of the Preference page. And also the *Generate void* should be check. Adjust the parameters if necessary. OK to exit.
- Click *File->Check before convert*, if no error, then a message will be given. Be sure to check the output message in the command window.





```
O######################O
O####CHECKING START####O
        CHECK NewComponent: OK!
                CHECK Berylium: OK!
                        CHECK Berylium1-Part1:   Volume:        1.21228e+07    I
NFO: This Part is not decomposed, will be done during conversion.
                CHECK CoolingPlate: OK!
                        CHECK CoolingPlate1-Part1:       Volume:        1.77012e
+06             INFO: This Part is not decomposed, will be done during conversio
n.
                CHECK Lithium: OK!
                        CHECK Lithium1-Part1:    Volume:        3.5274e+06     I
NFO: This Part is not decomposed, will be done during conversion.
O####CHECKING ENDED####O
O######################O
```

- Click *File->Convert to MC input*, select the location and give a file name, and *Save*.
- If success, a message will be given. as following:

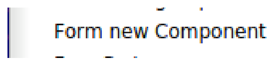
- If McCad does not response for a long time, check the command window for the output message, and see if problems.

4.2 MCNP6 hybrid CSG and mesh conversion

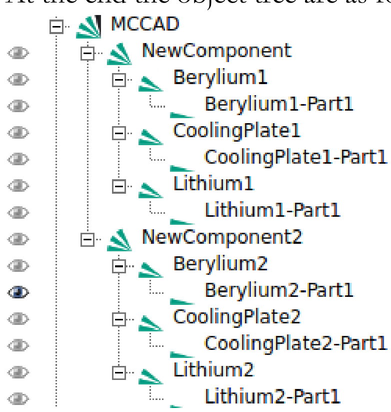
This tutorial demonstrates how to generate MCNP6 hybrid geometry input file. In McCad the target MC code option is “MCNP6”.

Step 1. Import geometry

- Import all the six CAD files: Berylium1.stp, CoolingPlate1.stp, Lithium1.stp, Berylium2.stp, CoolingPlate2.stp and Lithium2.stp.
- Choose Not putting in one group, and *explode* the compound.
- Select group Berylium2, CoolingPlate2 and Lithium2, right-click and choose *Form new Component*, give a name to this new component. Name this component as NewComponent2

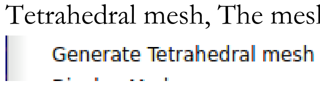


- At the end the object tree are as follow

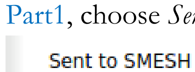


Step 2. Generate mesh

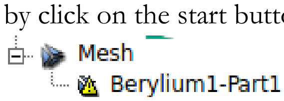
- In the Preference page, select *Tetgen* as meshing engine.
  - Selecting *Tetgen* creates so-call TT mesh
  - *Netgen* are not selected because it generates very coarse Finite element mesh, and we are not enable to adjust the size.
- Select Berylium1-Part1, Lithium1_Part1 and CoolingPlate1-Part1, right-click and choose Generate Tetrahedral mesh, The meshing will be finished in a few seconds.



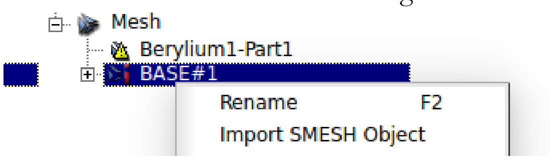
- Check the mesh by sending them to SMESH module. For example, right-click on the Berylium1-Part1, choose *Send to SMESH*



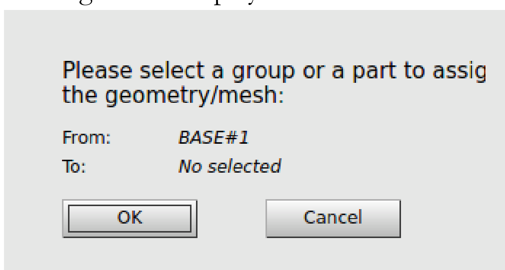
- Open the SMESH module by click on the start button , and the mesh are shown in the



- We can import a new mesh and replace the mesh McCad generated. In SMESH module, click *File->Import->CGNS* file, select file Berylium1_Mesh.cgns and open. The mesh is name as BASE#1 in SMESH, rename it if necessary.
- Back to McCad module by click on McCad start button, or right-click on any McCad object then choose *Activate MCCAD module*. Right-click on the BASE#1, and choose *Import SMESH object.*



- A dialog will be displayed. Choose which McCad part you want to assign the mesh and click OK.

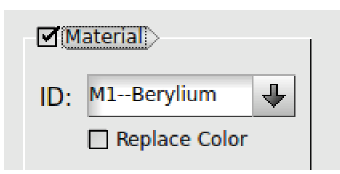


- Do this mesh imports similarly for CoolingPlate1-Part1 and Lithium1-Part1.

Step 3. Assign material.

- Open material management window from menu McCad->Edit Material, click *Import* to and choose material.xml file.
- Modify the data if needed. Then Click *OK* or *Cancel* to exit.
- Right-click on the group Berylium1 and Berylium2, and choose *Set properties* to show the Properties window

- Set the material to be M1--Berylium, and select a color for this group, and then click Apply.



- Similarly for CoolingPlate1, CoolingPlate2, and Lithium1, Lithium2.

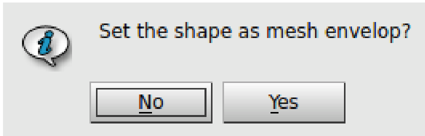
Step 4. Assign envelop

- Create a envelop for NewComponent which necessary for MCNP6 unstructured meshes. It means that all the Parts in this component are model with meshes. Right-click NewComponent and choose *Send to GEOM.*
- Then open the GEOM module by clicking on the GEOM start button (as highlighted). We can find the sent NewComponentare under the object tree of GEOM.

- In GEOM module, click *Measure->Dimensions->Bounding Box*, click *Apply* and *Close* for create a new bounding box.

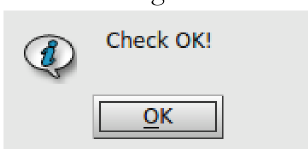
- Back to McCad module by click on McCad start button. Right-click the Bounding Box_1, and select *Import GEOM object*.

- A dialog will show up. select the NewComponent, and click OK. Anther Dialog will show up and ask if assign it as envelop for the component, we choose Yes.



Step 5. Conversion

- In the Preference page, select *MCNP6* as target MC code. And Check the Length conversion factor if correct.
- Click *File->Check before convert*. If no error, the following message will be given. In any case, check the screen message for this checking because the volume comparison of CAD and mesh will be given.




```
O#######################O
O####CHECKING START####O
        CHECK NewComponent: A meshed component.   OK!
                CHECK Berylium1: OK!
                        CHECK Berylium1-Part1:   Volume:       1.21228e+07
Mesh Volume: 1.21232e+07                Diff.: 3.30596e-05             OK!
                CHECK CoolingPlate1: OK!
                        CHECK CoolingPlate1-Part1:       Volume:        1.77012e+06
Mesh Volume: 1.76722e+06                Diff.: 0.00163817              OK!
                CHECK Lithium1: OK!
                        CHECK Lithium1-Part1:    Volume:       3.5274e+06
Mesh Volume: 3.52198e+06                Diff.: 0.00153706              OK!
        CHECK NewComponent2: OK!
                CHECK Berylium2: OK!
                        CHECK Berylium2-Part1:   Volume:       1.21228e+07        I
NFO: This Part is not decomposed, will be done during conversion.
                CHECK CoolingPlate2: OK!
                        CHECK CoolingPlate2-Part1:       Volume:        1.77012e+06I
NFO: This Part is not decomposed, will be done during conversion.
                CHECK Lithium2: OK!
                        CHECK Lithium2-Part1:    Volume:       3.5274e+06         I
NFO: This Part is not decomposed, will be done during conversion.
Comparison from CAD volume and Mesh volume      1.74203e+07     1.74124e+07      Dif
f:      0.0454688%
O####CHECKING ENDED####O
O#######################O
```

- Click *File->Convert to MC input,* select the location and give a file name, and Save.
- If success, a message will be given. as following:

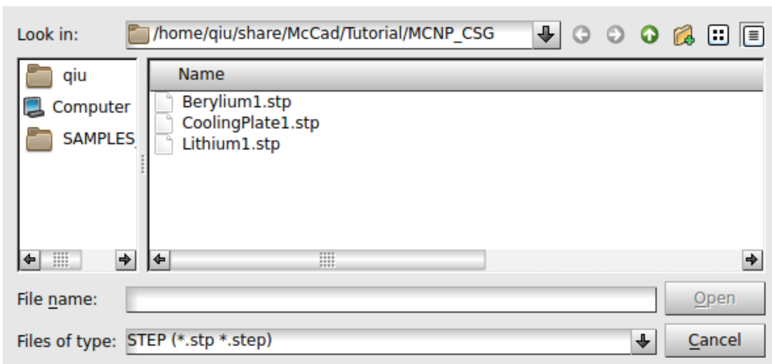
- The mesh file will be named with "*.inp" after export.

4.3 GDML half-space solid and tessellated solid conversion

This tutorial shows you how to generate a GDML file for an extended version of Geant4 code. There are two possible approaches, one is converting CAD solids to the half-space solids, and another is converting them into tessellated solids. You can use both solid types in one GDML file if you like.

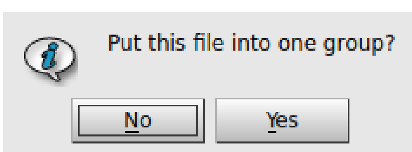
Step 1. Importing the geometry.

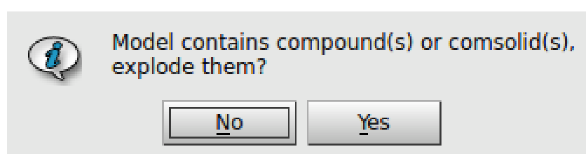
- Open menu *File->Import CAD file*, select files type to be STEP (default), and select all the three STEP files and open: Beryllium1.stp



- A dialog is shown and ask whether put CAD solid in these file into one group, we choose *No*.
- Then it will ask if explode the compounds. We choose *Yes* to explode them.

*NOTE: Sometimes only one solid is shown after explode, it might because the compound contains free faces or edges which have been filtered, or the compound just contain one solid.

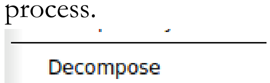




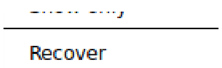
- At then end the object tree looks like following. We can right-click and show the geometry.

Step 2. Decompose the solids (converting half-space solid).

- Right-click the "NewComponent", and click *Decompose*, all the parts in this component will be decomposed. If this step is skipped, all the parts will still be decomposed during the conversion process.



- If the decomposed solid is too complex, we can right-click the part "Berylium1-Part1" and *Recover* the solid and do manual cut on it first.

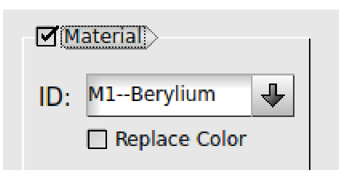


- If you want to convert these solids to tessellated solids, you don't need to do this step.
- Note: When solids are decomposed into Half-space solid, they will be merged again as Boolean Union Solids, which is actually not efficient for Geant4 simulation. We are going to use Multi-union solid in the future. But now you may manually avoid using the Boolean Union Solid by select all the Parts, right-click->Explode Parts, and then they will be export as Half-space solid.
- Note: You can check if the solids are decomposed by Click *File->Check before convert*. If you find some solids which are failed in decomposition, you can make interactive decomposition by following the Section 4.4.

Step 3. Create and assign material.

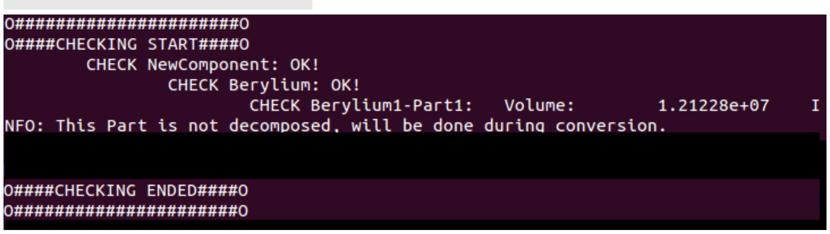
- Open material management window from menu McCad->Edit Material.
- Fill the "ID" (muss be unique) with 1, "Density" with 1.18 (g/cm^3), "Name" (muss be unique) with "Berylium", ignoring other items. Click *Apply* to create the material.
- Right-click on the group Berylium1, and choose *Set properties* to show the *Properties* window

- Set the material to be M1--Berylium, and select a color for this group, and then click Apply.

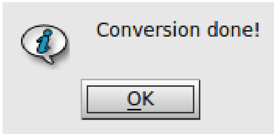


Step 4-1. Conversion to half-space solid

- First check if the *GDML* option is selected in *Preference ->Target MC code*. Adjust the parameters if necessary. *OK* to exit.
- Click *File->Check before convert*, if no error, then a message will be given. Be sure to check the output message in the command window. Note: please take care of the solids which are not decomposed. If you already decomposed them, then it means they are failed in decomposition.



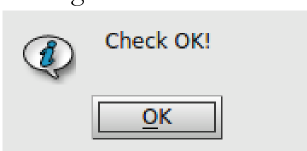
- Click *File->Convert to MC input*, select the location and give a file name, and *Save*.
- If success, a message will be given. as following:



- If the GUI does not response for a long time, check the command window for the output message, and see if problems.

Step 4-2. Conversion to tessellated solid (or both tessellated/half-space solid)

- Select part Berylium1-Part1 in the object tree, Right-click and choose *Mark as Undecomposed* or *Recover*. These parts will be considered as undecomposed parts and will be converted to tessellated solids.
- In the *Preference* page, uncheck the *General conversion -> Force decompose*. This prevents solids to be decomposed automatically. Click *OK* to exit.
- Click *File->Check before convert*, if no error, then a message will be given. Be sure to check the output message in the command window. Disregard those messages on undecomposed solid.



- Click *File->Convert to MC input*, select the location and give a file name, and *Save*.

Step 5. Use the GDML file in Geant4.

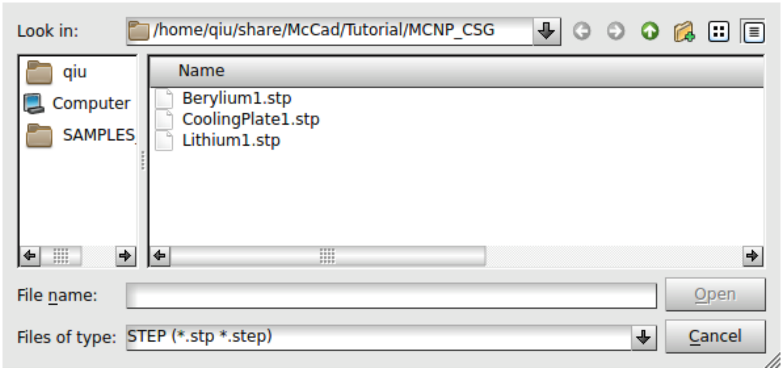
- First confirm that you have install correctly the half-space solid and extend version of GDML. Please read the half-space solid installation guide offer in the package https://github.com/Derek-yfqiu/Geant4-Halfspace-solid.
- First you have to change the GDML to find the correct xml schema. Open the GDML file, find the second line for the "gdml.xsd", and change it to the correct path, for example, "/opt/GEANT4/geant4.10.00/source/persistency/gdml/schema/gdml.xsd".
- Also you have to complete the material definition. In the Material block of the GDML file, the materials defined in the GUI is given, but not fully working. You have to complete them manually.
- At the end, load the GDML file in the Geant4 in the same way as loading other GDML file in your Geant4 application.

4.4 Interactive decomposition.

Sometimes the decomposition fails due to the reason of either CAD geometry problem or the failure of CAD Boolean operation engine. Therefore, it requires you to handle the decomposition interactively. In this tutorial, an example is given for this purpose.

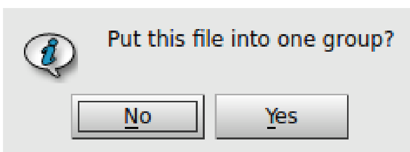
Step 1. Import the problematic CAD solid

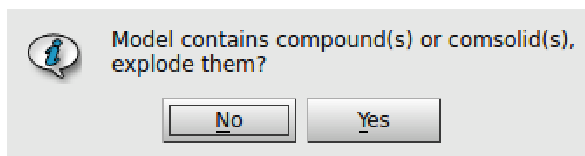
- Open menu *File->Import CAD file*, select files type to be STEP (default), and select all the three STEP files and open: CoolingPlate1.stp



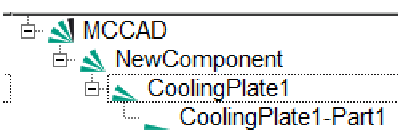
- A dialog is shown and ask whether put CAD solid in these file into one group, we choose *No*.
- Then it will ask if explode the compounds. We choose *Yes* to explode them.

*NOTE: Sometimes only one solid is shown after explode, it might because the compound contains free faces or edges which have been filtered, or the compound just contain one solid.



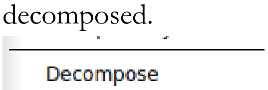


- At then end the object tree looks like following. We can right-click and show the geometry.

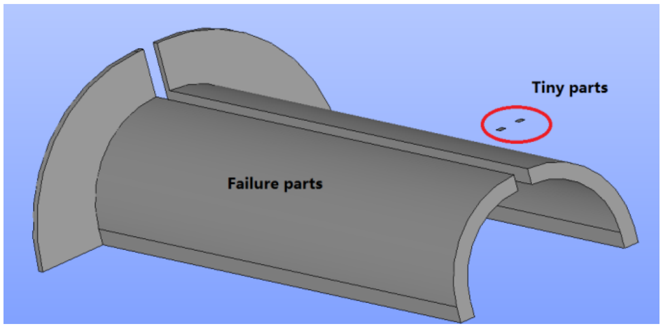


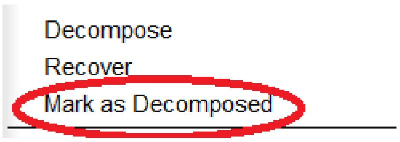
Step 2. Decompose the solids

- Right-click the "CoolingPlate1-Part1", and click *Decompose*, all the parts in this component will be decomposed.



- Click *File->Check before convert*, you will find this solid is not decomposed. To check which parts is failed in decomposition, you can right-click the "CoolingPlate1-Part1" and then *Explode parts*. Select all the exploded parts (or the group), right-click and then *Decompose*, and then Click *File->Check before convert* again.
- You will find out these parts are failed in decomposition. There are some tiny parts generated by the decomposition process. If you think they are not necessary, just delete them by right-click and *Delete Parts* . If you think they have to be considered, right-click and *Mark as decomposed* .

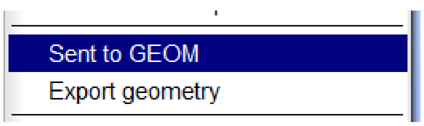


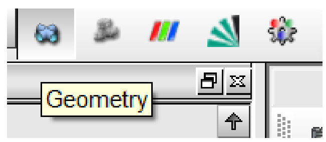


- Note: When use *Mark as decomposed* , you should confirm that they are sign-constant (half-space) solid which is able to be built by Boolean intersection of their boundary surface. Otherwise there might be geometry error!

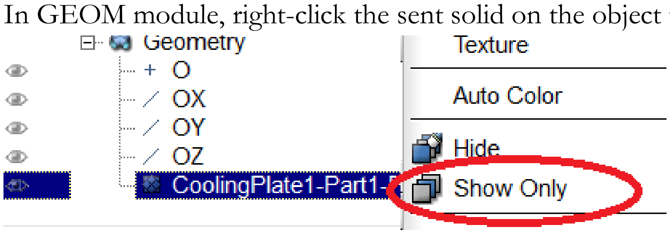
Step 3. Manual decomposition

- For the failure parts, you can try SALOME GEOM module to cut them. Right-click on the parts, and *Send to GEOM* , then activate the GEOM module by click the GEOM button.

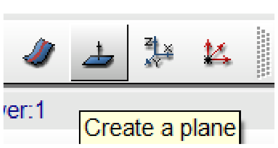


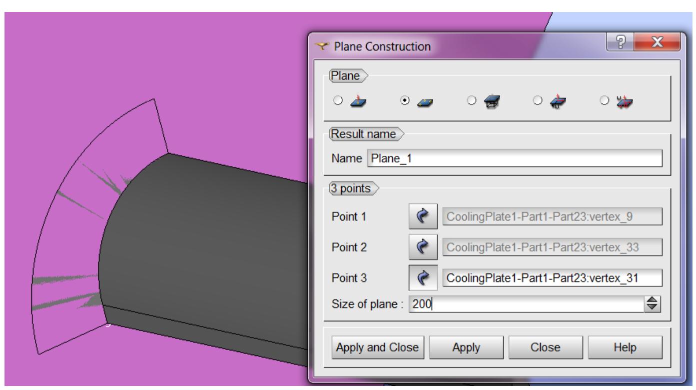


- In GEOM module, right-click the sent solid on the object three, and *show only* .

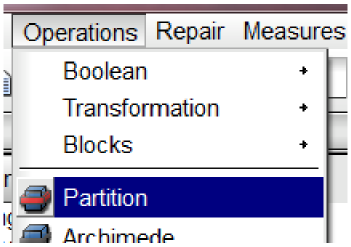


- Construct a plane for cutting the solid. You can create a plane by selecting three points from the solid.

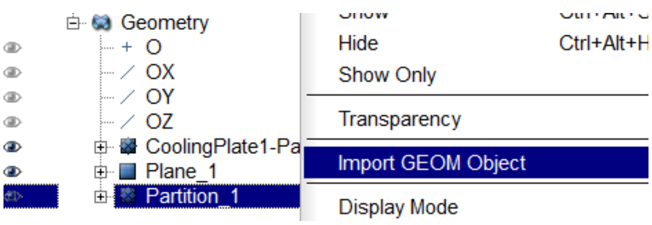


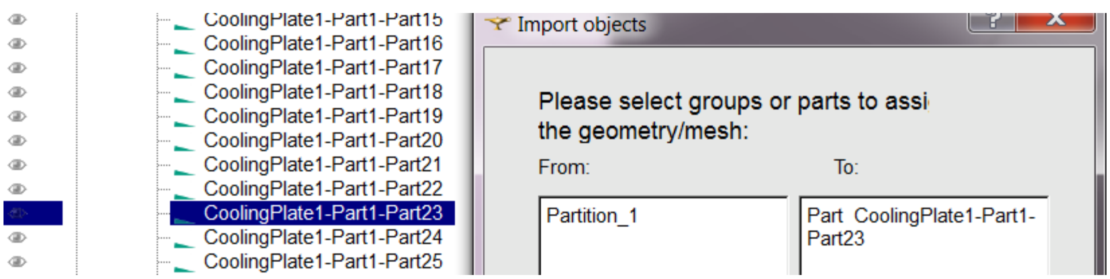


- Cut the solid using the *Operation* (menu)-> *Partition* . Select he failure solid as "Object", select the created plane for "Tool Objects", and *Apply and Close.* A new solid called "Partition_1" will be created.



- Send the solid back to McCad module and replace the previous solids. Switch to McCad module by clicking the button, right-click the "Partition_1" and then *Import GEOM Object* . A dialog is showing, and then you select the failure part in the McCad object tree. Click *OK,* the failure part will be replaced with the "Partition_1".





- Right-click this part again, and *Mark as Decomposed* if you are sure it has been fully decomposed. Otherwise you *Decompose* it again and check if success.