McCad user guide

Yuefeng Qiu, Lei Lu

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 MCNP (CSG and mesh geometry), TRIPOLI and Geant4 (CSG and tessellated solid). McCad has been integrated in the open-source SALOME platform, and relies it on user interfacing. This manual is written for giving a practical guide on how to generate an input file for these MC codes using McCad.

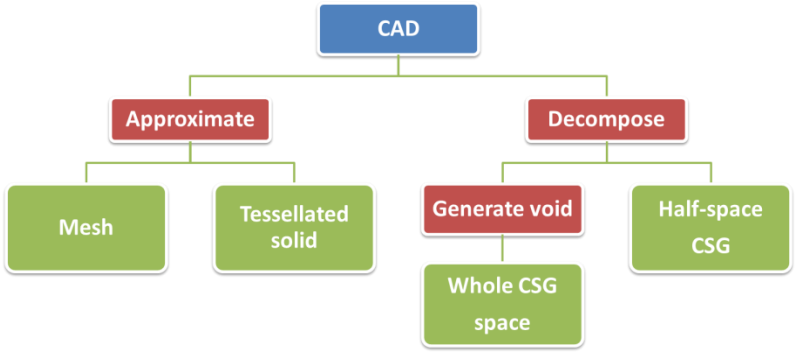


Fig 1.1 McCad workflow

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

- Decomposing process: CAD model are decomposed into finite number of solids which can be described as so-called sign-constant solids. These solids are formed by intersections of Boolean half-spaces;
- 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.
- 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 discretize the CAD geometry using mesh elements. McCad uses open-source tools to generate these data.

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. Only to keep in mind is that the void generation 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.
- 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.

2. Graphic user interface

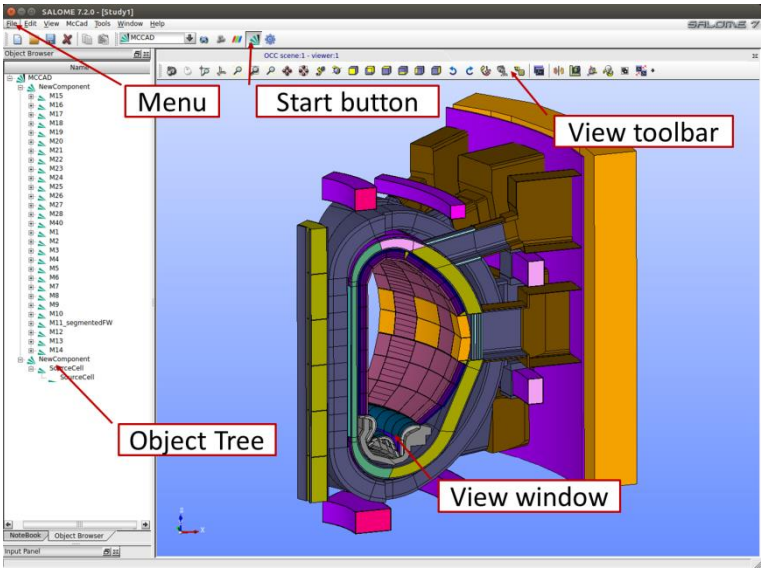


Fig 2.1 McCad GUI

Fig 2.1 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.

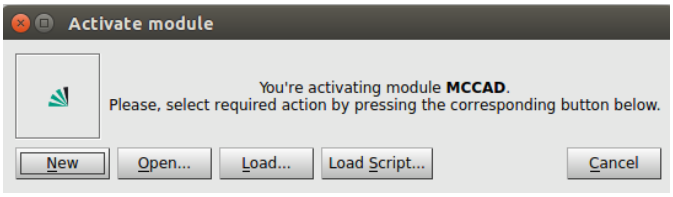
3. Detail guides

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: 

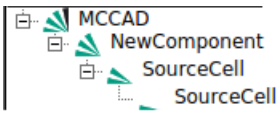
- then the following dialog pop-up:



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

3.2 Object tree

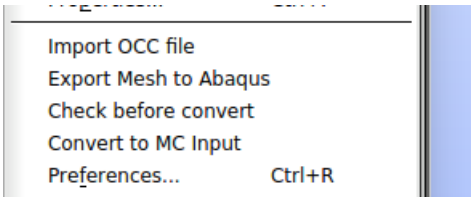
The object tree are organized as following :



- 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 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 can be assigned for these groups.
- Part: A part is an end node of the object tree. It represents a geometry entity that have only one material composition. It might contain just one CAD solid, or several CAD solid (we say a Compound), or CAD solids with an unstructured mesh.

3.3 File menu

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

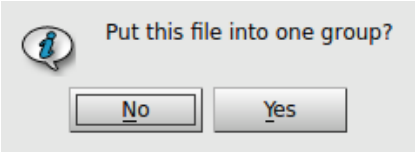


- Import OCC file: importing Open CASCADE CAD files, including BRep, STEP, IGES format.
- Export Mesh to Abaqus: export meshes to Abaqus mesh format. Mesh files in this format can be used for MCNP6 unstructured mesh geometry.
- Check before convert: Check the model before been converted into the MC input file. The program will conduce necessary check on the validity of the model, and all the message will be shown in the command window where SALOME is started.

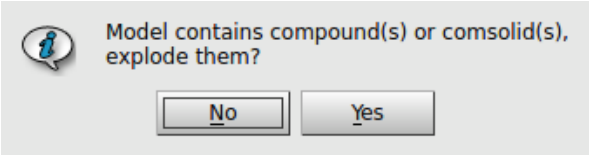
- Convert to MC input: converted the model into an input file of a chosen MC code. The target MC code can be chosen from the **Preferences** . Before conversion, an automatic check on the model will be performed.
- Preferences: see section in below.

3.3.1 Import OCC 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 would be better not putting CAD solids in different file into one group, order to distinguish the data.



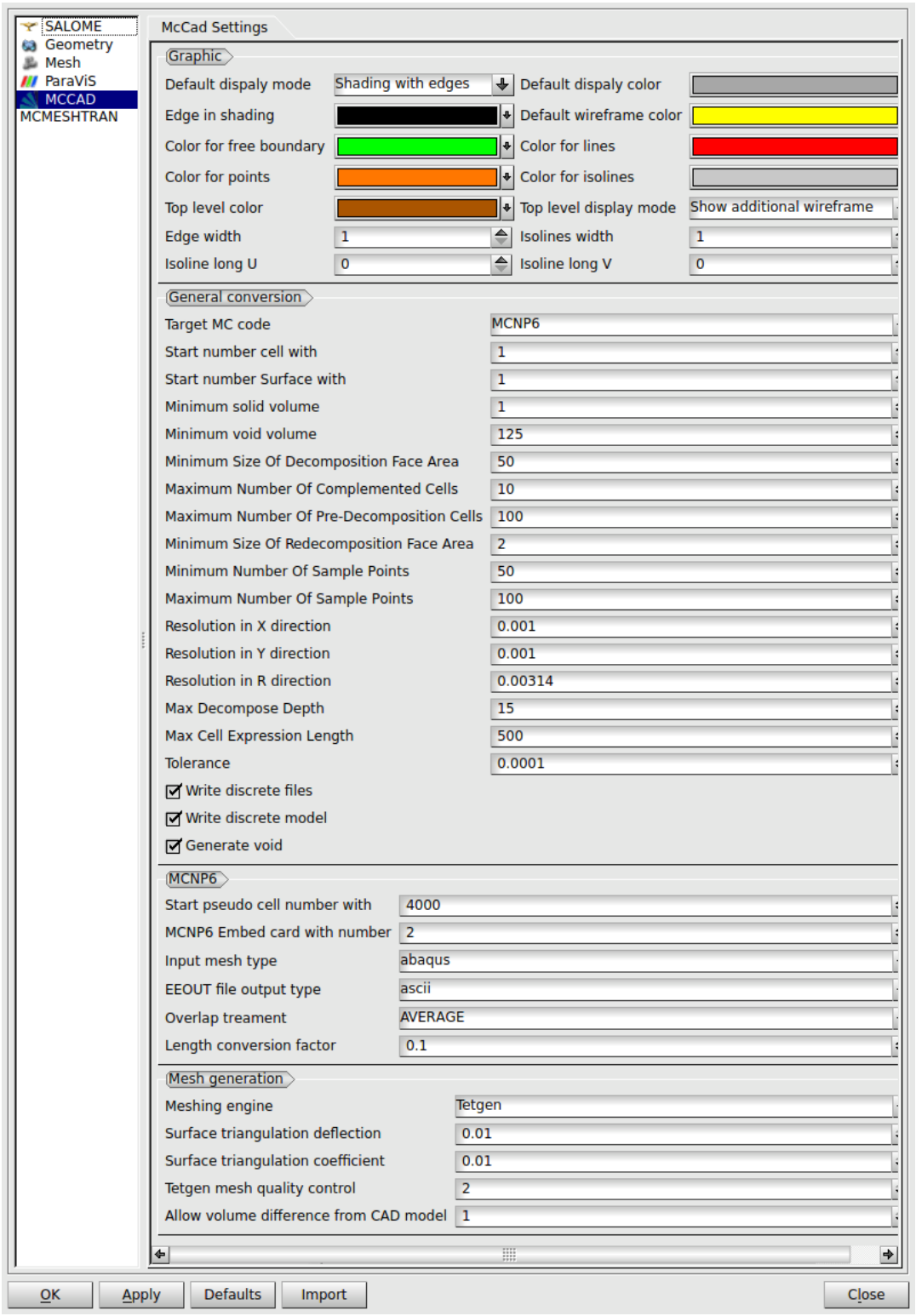
- If a file contains more than one CAD solid (or, contains a Compound), then it will ask if explode it.
- User can explode a compound solid or fuse some solid as a compound later in the GUI, if the decision is changed later.
- For generating mesh using McCad's own meshing function, it requires to explode 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 any other case, the imported data will be put in a new component.

3.3.2 Preference page

The preference page manage all the parameters used for the conversion. Currently it is not well organized and well implemented. 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: ??
- Max cell expression length: ??
- Tolerence: ??
- 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)

- 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

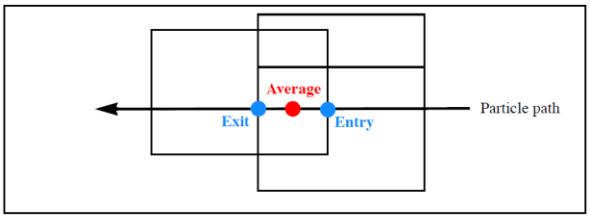
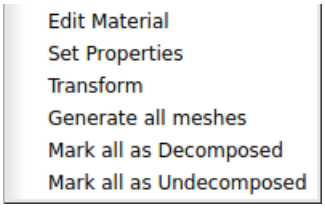


Figure: 3 - 1. Illustration of the three critical points for the overlap models.

    - 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 absolute tolerance in faceting the solid
  - Surface triangulation coefficient: set the relative tolerance in faceting the solid, the value is adjust according to the size of each solid
  - Allow volume difference from CAD model: Allowable difference between CAD volume and mesh volume. This value set limits for check if the mesh are consistent with the CAD solid.
  -

3.4 McCad menu

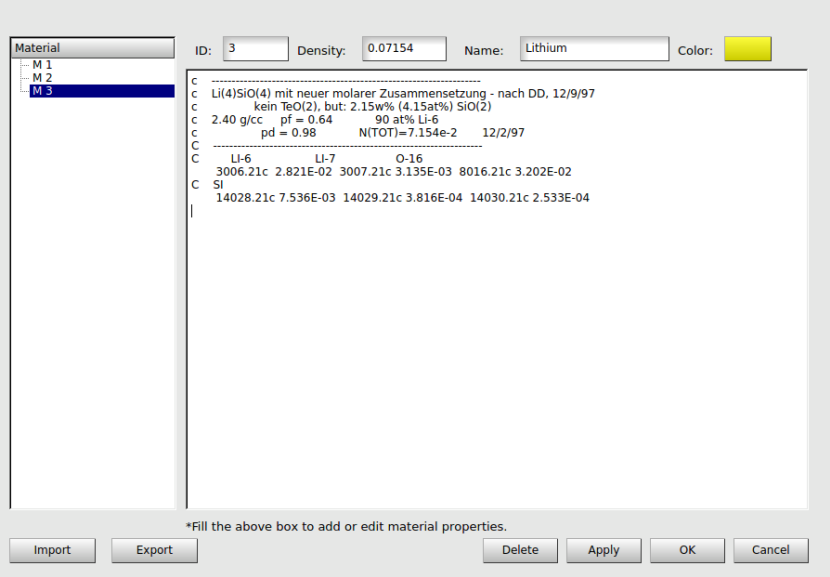
In the McCad menu, the following functions have been implemented:



- 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 are shown as following. Currently the material management is tested only for MCNP and TRIPOLI code.

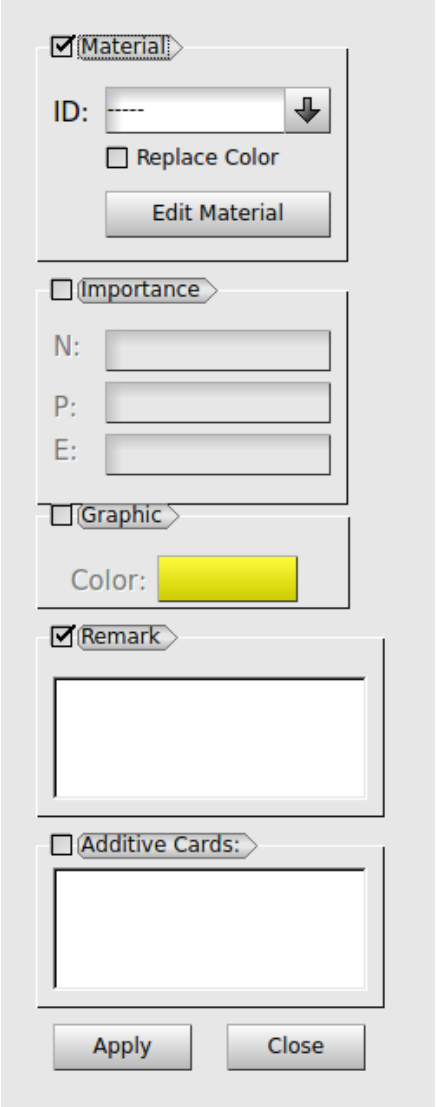


- Material tree: for displaying the material
  - The displaying name of the material is: M+ID
- ID: material identification number, should be unique. This is initially design for MCNP.
- Density: Atom density (positive value) or mass density (negative value)
- Name: Name for the material. Highly recommend 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

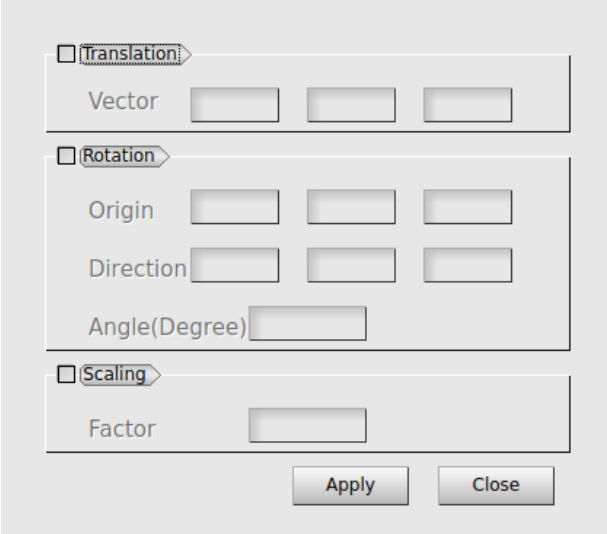
The properties of component/group/part can be set using the Properties dialog. This dialog is interactive with the object tree, and enable to assigned properties for multiple objects. The properties assigned for object in upper level will overwrite that of objects in lower level. The properties which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



- 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

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

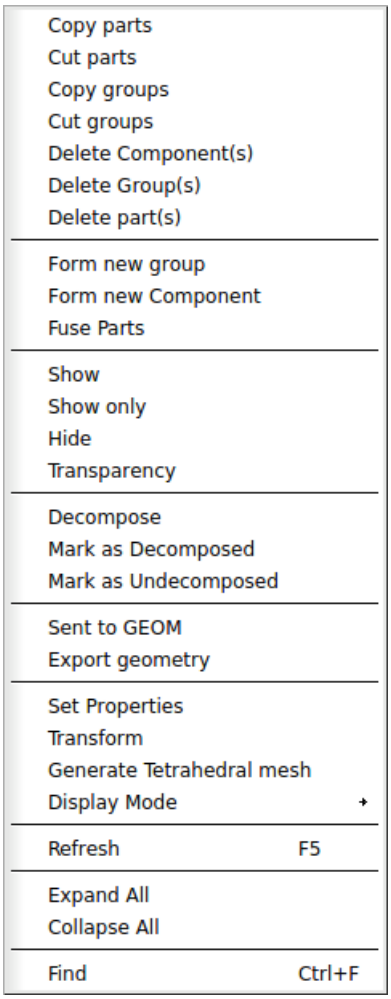


- Translation
  - Move the model according to the vector
  - Vector: X, Y, Z value of the vector
- Rotation
  - rotate the model according to the Axis and angle
  - Origin: origin point of the axis
  - Direction: direction vector of the axis
  - angle: rotate angle in degree
- Scaling
  - Scaling the 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
- 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. USE 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.
-  **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.
-  **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.
-  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

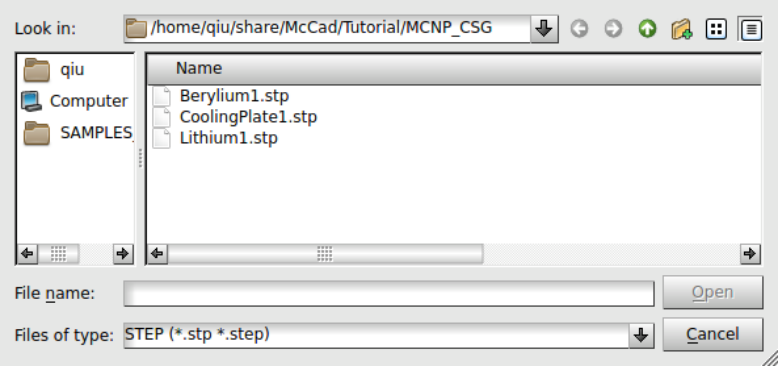
These tutorials provide practical example for showing how to create a input file for MC simulation. It need to be noticed that the input file create by McCad is not completed, therefore user should add necessary parameters before using it for calculations.

4.1 MCNP traditional CSG conversion

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

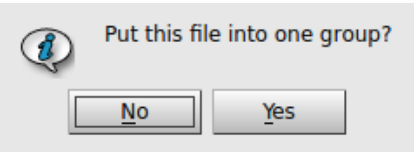
Step 1. Importing the geometry.

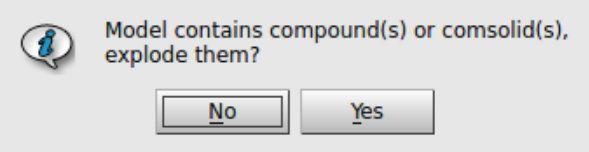
- Open menu File->Import OCC file, select files type to be STEP (default), and select all the three STEP files and open: Berylium1.stp, CoolingPlate1.stp and Lithium1.stp.



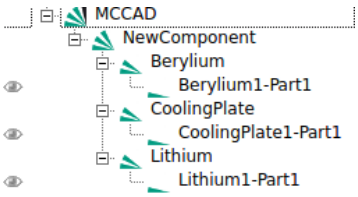
- A dialog is shown and ask whether put CAD solid in these file into one group, we choose No.
- Then it will ask if explode the compounds. We choose Yes to explode them.

*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.



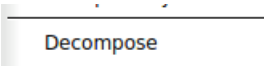


- At then end the object tree looks like following. We can right-click and show the geometry.
-

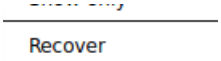


Step 2. Decompose the solids (optional).

- 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.

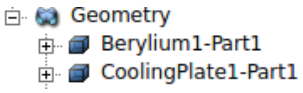


- If the decomposed solid is too complex, we can recover the solid and do manual cut on it first.

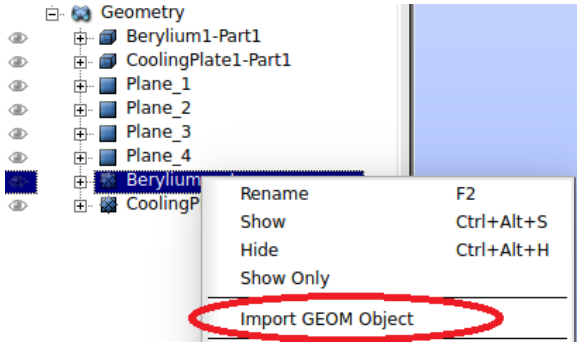


Step 3. Manual cut the solid (optional).

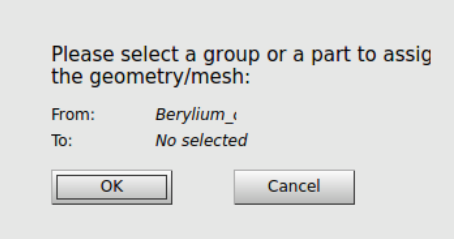
- Select the "Berylium-Part1" and "Lithium-Part1", right-click and choose *Send to GEOM*.
- Then open the GEOM module by clicking on the GEOM start button (as highlighted). We can find the sent geometries are under the object tree of GEOM.



- 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.

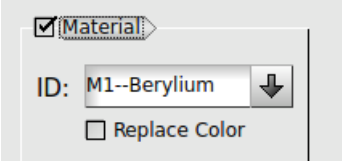


- Visualize the object in order to confirm the correctness.

Step 4. Assign material.

- Open material management window, 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

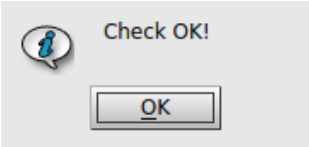
- Set the material to be “M1--Berylium”, and select a color for this group, and then click Apply.



- Similarly for “CoolingPlate” and “Lithium”.

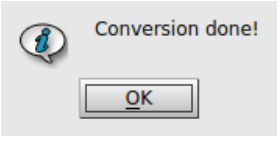
Step 5. Conversion

- First check if the “MCNP5” 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:



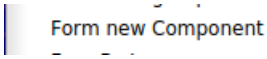
- If McCad does not response for a long time, check the command window for the output message, and see if problems.

4.2 MCNP6 hybrid CSG and mesh conversion

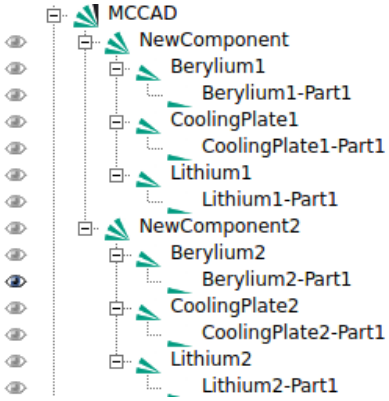
This tutorial demonstrates how to generate MCNP6 hybrid geometry input file. In McCad the target MC code option is “MCNP6”.

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”

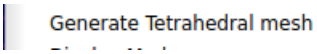


- At the end the object tree are as follow



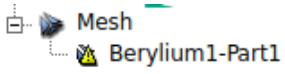
Step 2. Generate mesh

- 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.

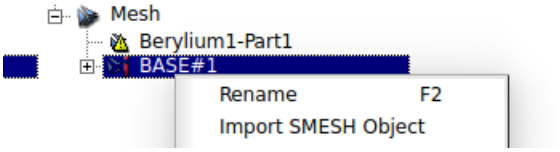


- Check the mesh by sending them to SMESH module. For example, right-click on the “Berylium1-Part1”, choose Send to SMESH

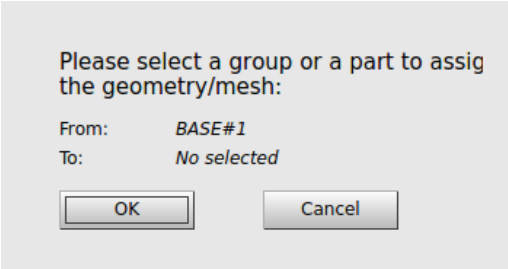
- Open the SMESH module by click on the start button , and the mesh are shown in the



- 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.



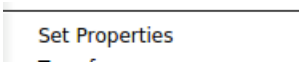
- A dialog will be displayed. Choose which Part you want to assign the mesh and click OK.



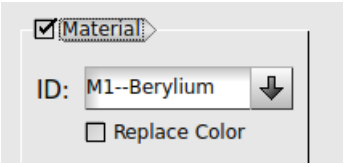
- Do this mesh imports similarly for “CoolingPlate1-Part1” and “Lithium1-Part1”.

Step 3. Assign material.

- Open material management window, 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.

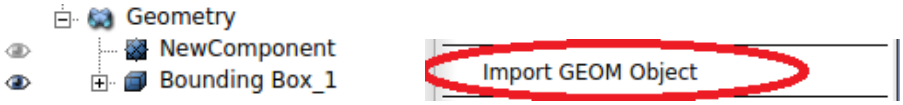


- Similarly for “CoolingPlate1”, “CoolingPlate2”, and “Lithium1”, “Lithium2”.

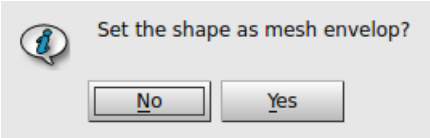
Step 4. Assign envelop

- 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 “NewComponent”are 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.

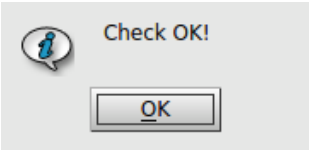


- A dialog will show up. select the “NewComponent”, and click OK. Anther Dialog will show up and ask if assign it as envelop for the component, we choose Yes.



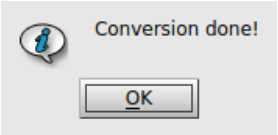
Step 5. Conversion

- 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.