# McMeshTran User Guide

Yuefeng Qiu

1. Overview

McMeshTran is a multi-physics coupling tool. It provides MC code mesh tally post-processing, mesh manipulation, data mapping and CFD/FE code interfacing. Currently it support post-processing MCNP mesh tally, MCNP6 unstructured mesh output, TRIPOLI-4 mesh tally and Geant4. It provides interfaces for CFD codes CFX and Fluent, FE code ANSYS Workbench.

The general workflow of McMeshTran is show as follow figure. The data post-processed from MC code are stored using a general mesh library MED. After necessary manipulations, e.g. summing, averaging, spatial transformation, etc., data are mapped into the target CFD/FE mesh on the consistent geometry. The data is then exported for CFD/FE codes then the multi-physics coupling simulation can be carried out.
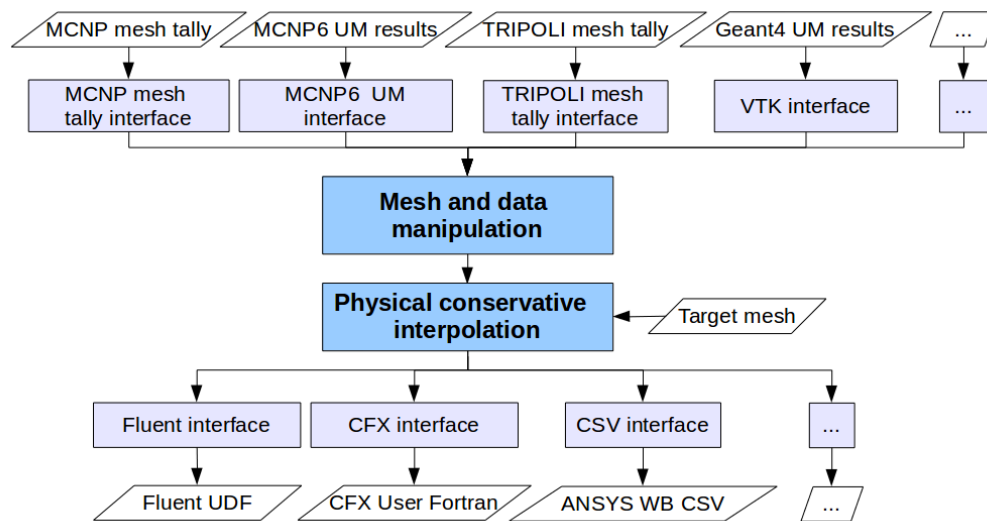
2. Graphic user interface

The following figure 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 *McMeshTran* menu have implementations functions for McMeshTran.
- **Object tree**: mesh objects.
- **View window**: mesh and data displaying window.
- **View toolbar**: interactive functions for mesh and data visualization.
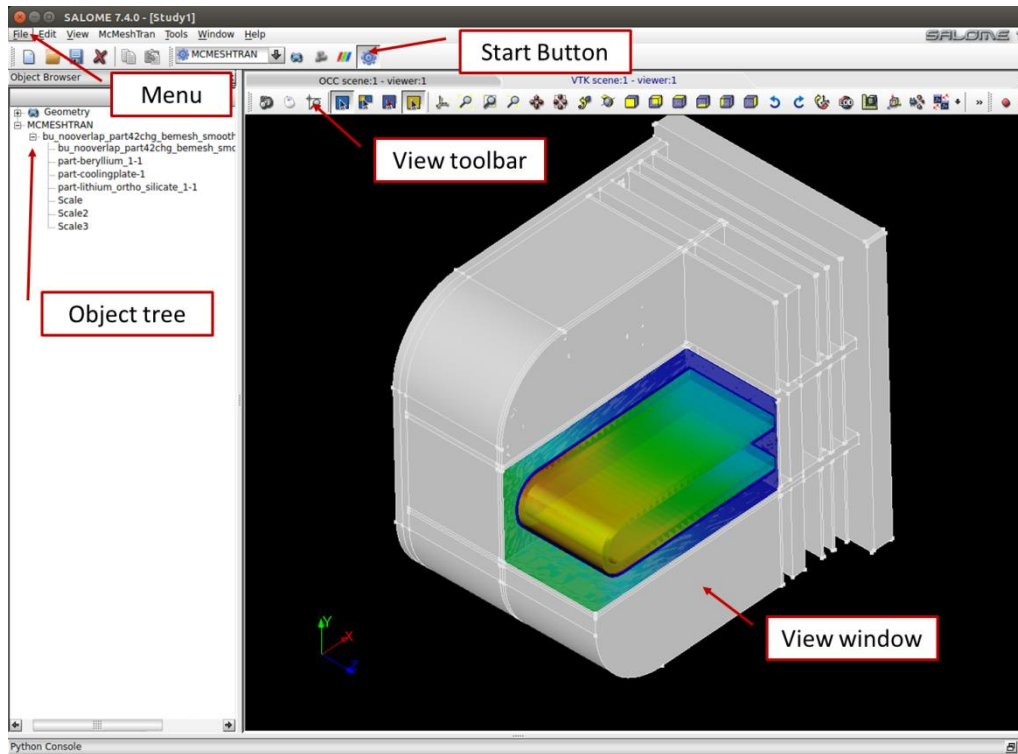
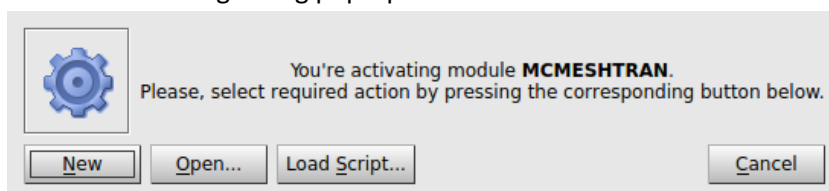# 3. Detail functions

3.1 starting McCad

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

- Start SALOME platform, click the McMeshTran **start button**:
- Then the following dialog pop-up:

You're activating module **MCMESHTRAN**.
Please, select required action by pressing the corresponding button below.

New   Open...   Load Script...   Cancel

- Click **New** for create a new McCad project.
- Then the McMeshTran GUI will show up.

3.2 Object tree

The object tree of McMeshTran is as follow.

- **MCMESHTRAN**: the root object. The root object is unique for each SALOME module.
- **Group**: Grouping mesh in need
- **Mesh**: Containing mesh, or mesh together with data.

3.3 File menu

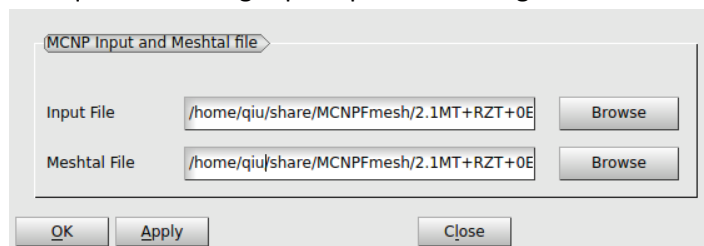The McMeshTran functions implemented in the *File* menu is as follow:

Import MC Data	▸
Import External Data	▸
Import External Mesh	▸
Export Data	▸
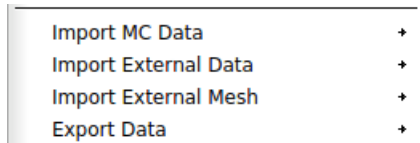
- *Import MC Data*: Post-processing MC output. Currently interfaces are offer for MCNP5, MCNP6 and TRIPOLI-4
- *Import External Data*: Import data from external mesh files. Currently support MED and CGNS format. Currently the CGNS interface is suitable for only one mesh in a file.
- *Import External Mesh*: Import mesh without any data. For importing target mesh. Currently supported format are CGNS, UNV, SAUV, STL, Abqaus (MCNP6 style).
- *Export Data*: Export mesh and data. Currently provide interface file for CFX and Fluent, and also other universal format, e.g. VTK, MED, CSV, CGNS, Abaqus (Mesh only)

3.3.1 Import MC Data

The Import MC data item have three sub-items, as following:

Import MC Data	◂	Read MCNP mesh tally
Import External Data	▸	Read MCNP6 UMesh eeout file
Import External Mesh	▸	Read TRIPOLI output file

- *Read MCNP mesh tally*: process MCNP mesh tally output. Both MCNP input and mesh tally output are required. A dialog is prompt when calling this function.

MCNP Input and Meshtal file		
Input File	/home/qiu/share/MCNPFmesh/2.1MT+RZT+0E	Browse
Meshtal File	/home/qiu/share/MCNPFmesh/2.1MT+RZT+0E	Browse
OK Apply		Close

  - Input file: MCNP input file which contain correct mesh tally parameter for the mesh tally file
  - Meshtal file: mesh tally output file.
- *Read MCNP6 UMesh eeout file*: processing MCNP6 unstructured mesh output file. The dialog is as follow:
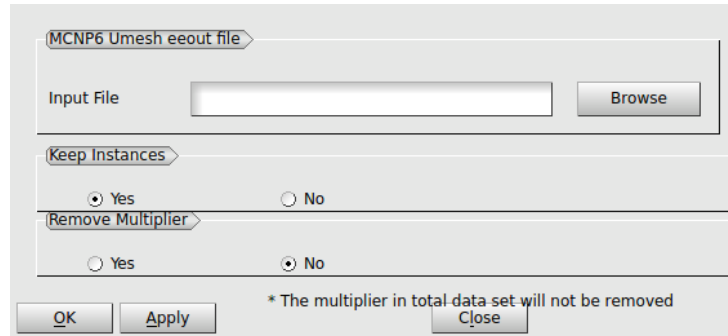
- o *Input file*: the unstructured mesh output file which need to be processed.
- o *Keep instances*: In MCNP6 output, the mesh are merge together as one mesh. McMeshTran can separate it into meshes for each instance. *Yes* to keep these instances, and *No* to get only the merged mesh.
- o *Remove Multiplier*: User can chose to remove the multiplier set for time and energy bins. If no multiplier is set in the *eeout* file, both options are equivalent. Noted that the multiplier in the total data set will not be removed.
- *Read TRIPOLI output file*: process TRIPOLI mesh tally result. Because all input and results data are given, only the output file is needed.

3.3.2 Import External data and mesh

- *Import MED*: Import MED mesh file. File can contain multiple mesh and data field, or only mesh without data.
- *Import CGNS*: Import CGNS mesh file. Mesh can only contain **one** CGNS Base and one Zone. Currently not support for MCNP6 results.
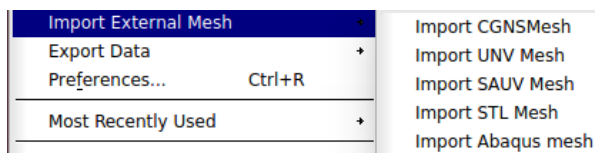
- *Import CGNS mesh*: only mesh in CGNS format.
- *Import UNV mesh*: mesh in Ideas UNV format.
- *Import SAUV mesh*: CEA Cast3M mesh
- *Import STL mesh*: only surface mesh.
- *Import Abaqus mesh*: Abaqus mesh file in MCNP6 style.

3.3.3 Export data

One should select a mesh object from the tree before export.
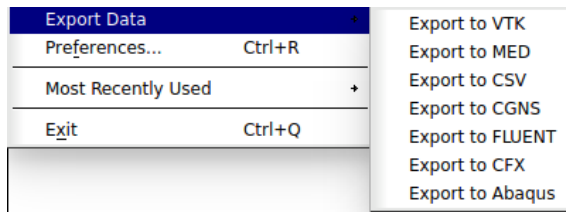
- *Export to VTK*: export to VTK format for visualization
- *Export to MED*: saving mesh and data in MED format
- *Export to CSV*: export to CSV format, which data are separated by comma or other delimiter. Noted that only points will be kept, either node or cell-centroid. No mesh information will be kept.
- *Export to CGNS*: export to CGNS format. Currently not support MCNP6.
- *Export to Fluent*: export data as heat source for Fluent code. A *C* source file will be generated and need to be compiled in Fluent code.
- *Export to CFX*: export data as heat source for CFX code. A Fortran source file will be generated and need to be compiled in CFX code.
- *Export to Abaqus*: export only mesh to Abaqus format.

3.4 McMeshTran menu

These functions are provided for manipulating the meshes and results. Meshes can be structured or unstructured mesh. Currently most of them do not support MCNP6 results.

3.4.1 Sum meshes
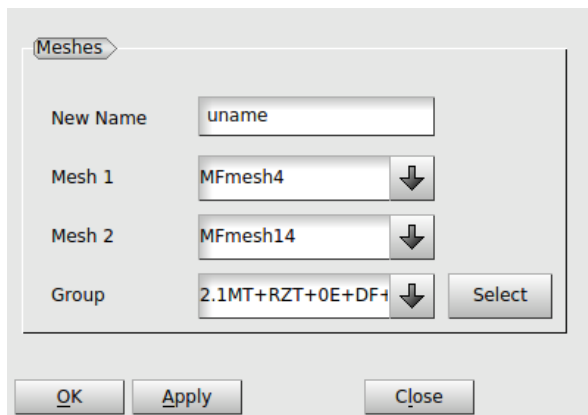
Summing the mesh tally results, e.g. neutron and photon heating. Currently do not support MCNP6.
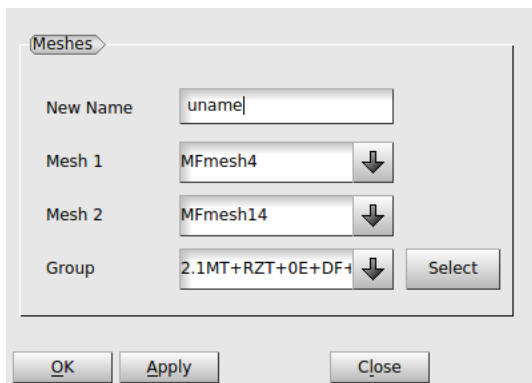
- *New name*: name for the results
- *Mesh 1*: mesh with results
- *Mesh 2:* should have the same mesh with Mesh 1.
- *Group*: put the results in which group.
- *Select*: select the group from object tree
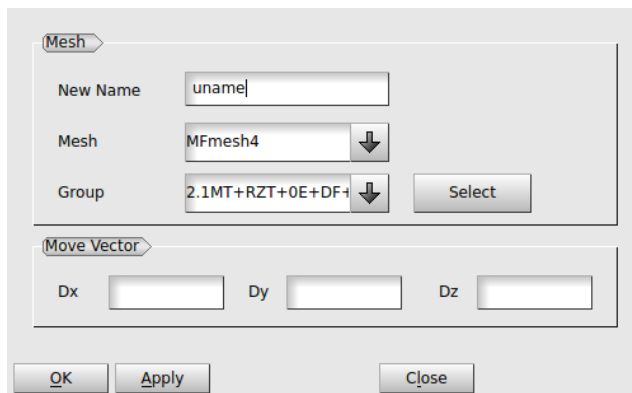
3.4.2 Average mesh

Averaging the mesh tally results, e.g. get an average results from two calculations. Currently not support MCNP6.

- *New name*: name for the results
- *Mesh 1*: mesh with results
- *Mesh 2*: should have the same mesh with Mesh 1.
- *Group*: put the results in which group
- *Select*: select the group from object tree

3.4.3 Translate mesh

Moving the mesh according to a vector.

- *New name*: name for the results
- *Mesh*: mesh with results
- *Group*: put the results in which group
- *Select*: select the group from object tree
- *Move vector(Dx, Dy, Dz):* Distance to move in each direction

3.4.4 Rotate mesh

Rotate the mesh according to an axis and an angle.

- *New name*: name for the results
- *Mesh*: mesh with results
- *Group*: put the results in which group
- *Select*: select the group from object tree
- *Center*: origin point of the axis
- *Vector*: direction vector of the axis
- *Angle*: rotate angle in **degree**.

3.4.5 Scaling mesh

Scaling the mesh referring to the origin point (0,0,0). Use it for converting the dimension unit.
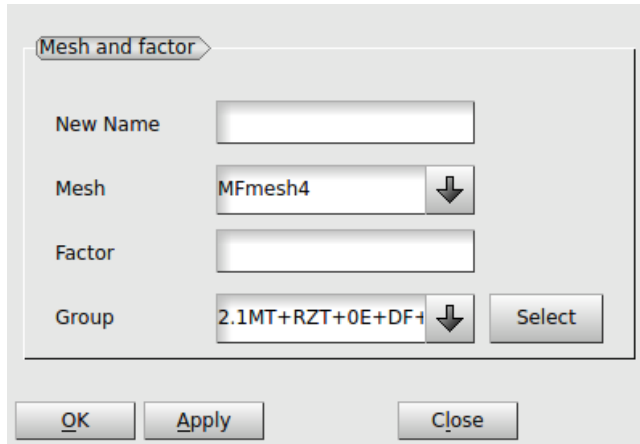
- *New name*: name for the results
- *Mesh*: mesh with results
- *Factor*: scaling factor.
- *Group*: put the results in which group
- *Select*: select the group from object tree

3.4.6 Multiple factor

Multiplying the results with a constant factor. Useful for converting the unit of the results.

- *New name*: name for the results
- *Mesh*: mesh with results
- *Factor*: constant factor.
- *Group*: put the results in which group.
- *Select*: select the group from object tree

3.4.7 Interpolate mesh

Interpolate the results on one mesh into another mesh. Here are some requirements for the interpolation in order to get a correct interpolation.

- Target mesh should be consistent with the Source mesh. In case of difference in units, check them by visualization. User should take care of the results if the geometry is not consistent.
- Volumetric mesh to volumetric mesh. Surface mesh interpolation are not supported.
- Check the results after interpolation. No explicit error checking provided by the program.

The dialog is shown as follow.

- *New name*: name for the results
- *Source Mesh*: mesh with results
- *Target Mesh*: The mesh where the data will be mapped.
- *Group*: put the results in which group.
- *Select*: select the group from object tree
- *Interpolate Tools*
  - *CGNS Utility*: USE WITH CAUTION! The interpolated method are based on Point-to-Point scheme, and should be careful if interpolate results on cells;
  - *MED Utility*: MED library function. ONLY tested for cell-to-cell interpolation.
- *Solution location*
  - *Cell-center*: locate the results on cells. Use it for Finite Volume Method based CFD codes.

o   *Vertex*: locate the results on mesh nodes (points). Use it for Finite Element method based codes.

3.5 Pop-up menu

The pop-up menu appears when right-clicking one or multiple objects in the object tree and the view window. It varies when difference objects are selected and the place they are selected.

- *Rename*: rename the object
- *Copy*: copy the object
- *Cut*: cut the object
- *Paste*: paste the object in selected place
- *Delete*: remove the object
- *Create group*: create a new group

- *Display*: show the object. The result will be visualized with default linear color map. If only mesh, the mesh will be visualized with silver color.
- *Erase*: hide the object
- *Transparency*: set the transparency
- *Display in ParaView*: Send the object to ParaView. ATTENTION: ParaView module should be activate before using this function. Click this button to activate ParaView:
- *Send to SMESH*: Send the mesh (without data) to SMESH module.
- *Operation*: mesh and data manipulation. Varies depending on the object selected
  - Translate mesh: See Section 3.4.3. One mesh selected.
  - *Rotate mesh*: See Section 3.4.4. One mesh selected.
  - *Scaling mesh*: See Section 3.4.5. One mesh selected.
  - *Multiply factor*: See Section 3.4.6. One mesh selected.
  - *Sum meshes*: See Section 3.4.1. Two meshes selected.
  - *Average meshes*: See Section 3.4.2. Two meshes selected.
  - *Interpolate mesh*: See Section 3.4.7. Two meshes selected.
- Display mode: visualization mode
  - *Point mode*: show the mesh node only
  - *Wireframe*: show the mesh edges
  - *Shading*: Show the faces,
  - *Shading with edge*: show faces with edges.
- *Export*: See Section 3.3.3.
- *Refresh*: update the object tree.
- *Find*: search for an object with string.
- *Dump View*: capture the screen and save as an image
- *Change Background*: change background color or image.

3.6 View Toolbar
These tools are provided by SALOME GUI. The functionalities of these tools are listed here 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.
- These buttons allow switching between three pre-selection (highlighting) modes:
  - Static pre-selection mode - pre-selection is done in terms of bounding boxes;
  - Dynamic pre-selection mode - pre-selection is done in terms of cells.
  - Disable pre-selection - pre-selection is disabled
- *Enable/Disable selection* - enables or disables selection in the view.

- *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
- *Change View Parameters* - this button gives access to the dialog for customization of various view parameters.
- *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.
- *Orthogonal mode* - Switches the view to the orthogonal mode.
- *Perspective mode* - Switches the view to the perspective mode.
- These buttons allow recording viewing operations as a video file in the AVI format using external software (jpeg2yuv)

-

4. Tutorials

## 4.1 MCNP mesh tally post-processing and multi-physics coupling

This tutorial will demonstrate how to post-processing MCNP mesh tally, and export the data for multi-physics coupling analysis. For import mesh tally results, McMeshTran needs the mesh tally configuration in the input file.

Step 1. Import mesh tally results

- Open *File->Import MC Data ->Read MCNP mesh tally*, a dialog will show as follow.
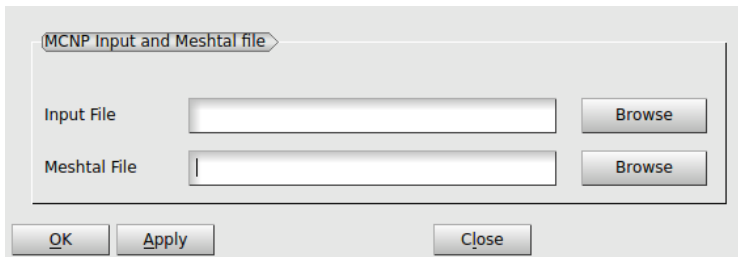
- In the *Input* File item, click Browse to load the MCNP input file: MCNP_input
- In the *Meshtal* file item, click Browse to load the mesh tally output: MCNP_meshtal
- Click *OK* to finish loading. A dialog will show the message "Operation Done.", and the object tree are as follow. It contains three mesh tallies
  - MFmesh4: neutron heating tally on Cartesian coordinate
  - MFmesh14: photon heating tally on Cartesian coordinate
  - MFmesh24: neutron heating tally on Cylindrical coordinate
  - MFmesh24: photon heating tally on Cylindrical coordinate
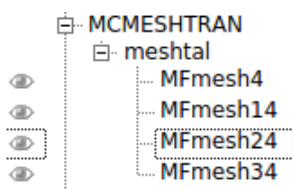

Step 2. Summing the mesh tally

- select MFmesh4 and MFmesh14, right-click and choose *Operation->Sum Meshes*, the following dialog will appear:
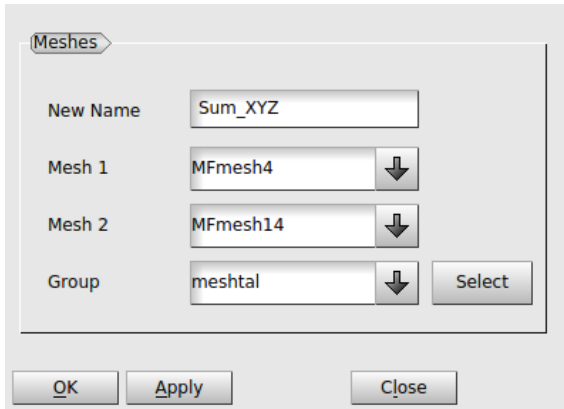
-
- Given an name "Sum_XYZ" to the results, and click *OK*.
- Similarly, summing up the MFmesh24 and MFmesh34 and name it as "Sum_RZT".

Step 3. Interpolate the results

- Select "Sum_XYZ" and "Sum_RZT", right-click and choose Interpolate Mesh, this dialog appears
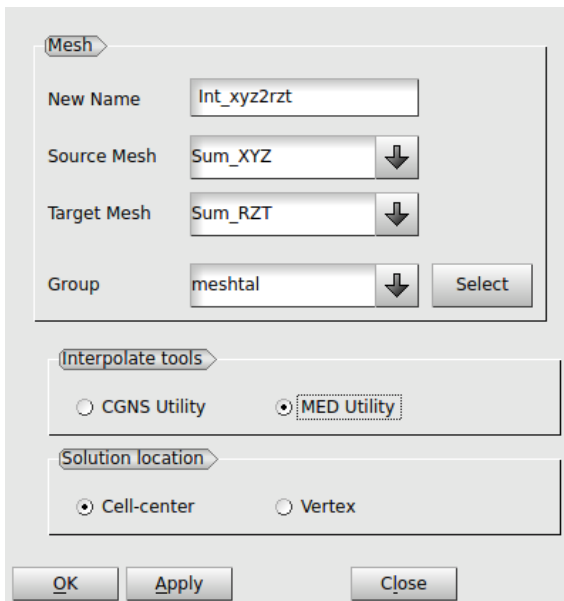
- Name the results as "Int_xyz2rtz", set "Sum_XYZ" as source mesh and "Sum_RZT" as target mesh, and choose *MED utility* and *Cell-center* for the interpolation option. Click *Apply*.
- Similarly, switch the "Sum_XYZ" to be target mesh and "Sum_RZT" to be source mesh, and name it as "Int_rzt2xyz".

Step 4. Visualize in ParaView

- Activate the ParaView module with its *start button*:
- Return to the McMeshTran, select "Int_xyz2rtz" and "Sum_RZT", right-click and choose *Display in ParaView*. Compare them for the results.
- Similarly send "Int_rzt2xyz" and "Sum_XYZ" to the ParaView, and compare them by visualization.

Step 5. Export for CFD codes

- In McMeshTran, right-click "Int_xyz2rtz" and choose *Export->Export to Fluent*. Give the name of the file with "*.udf".

# 4.2 MCNP6 unstructured mesh output Post-processing

This section will describe how to post-processing MCNP6 unstructured mesh output file, visualize the result together with the geometry. The use of MCNP6 unstructured mesh result for CFD analysis is currently not available, but it will be implemented very soon.

Step 1. Import unstructured mesh output

Open *File->Import MC Data ->Read MCNP6 UMesh eeout*,
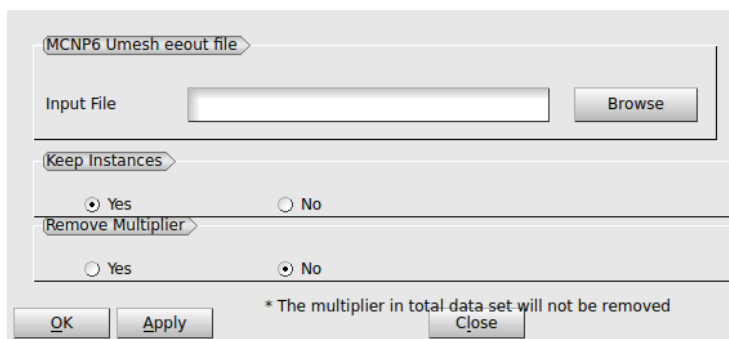
Click Browse to load the output file: /MCNP6_UM_EEOUT/Breeder_Unit.eeout. Keep the following option by default, *OK* to start processing. The object tree is as follow.
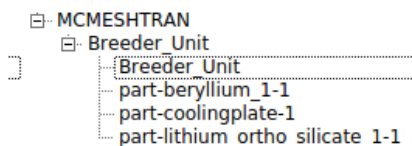
- Breeder_unit: The entire mesh contains all the unstructured meshes in the eeout file
- part-beryllium1-1: the beryllium mesh, matching with the instance of beryllium part-beryllium1-1 in the unstructured mesh file
- part-coolingplate-1: the mesh of cooling plate
- part-lithium_ortho_silicate_1-1: the mesh of lithium

Step 2. Convert the unit of heating to W/cm³

The results from F6 tally of unstructured mesh are in MeV/g, they need to be converted it to W/cm3. The conversion factor is calculated using : 1.60217733e-13 j/MeV * Source intensity * mass density in g/cm3. Using the *Multiply Factor* function and multiply the conversion factor for the three component as follow:
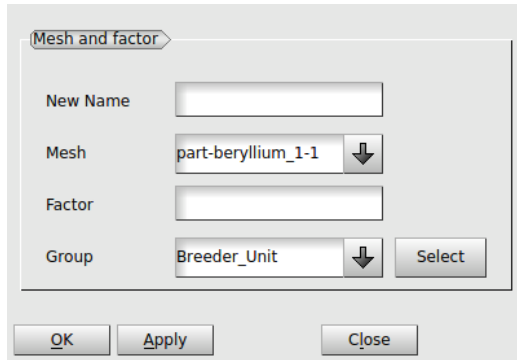
- part-coolingplate-1: New name: MT_Coolingplate, factor 1.35239E+05
- part-lithium_ortho_silicate_1-1: New name MT_Lithium, factor 2.63197E+04
- part-beryllium1-1: New name MT_Beryllium, factor 3.14331E+04

Step 3. Send to ParaView for visualization.

- Activate the ParaView module with its *start button*:
- Back to the McMeshTran Right-Click the MT_Lithium, choose *Display In ParaView*. The mesh and data will be sent to ParaView module.
-

Step 4. Visualize the result with the geometry

- In ParaView, click *Open* (Not the SALOME open icon), choose the CoolingPlate.stp.
- Adjust the deflection to increase model precision.

Appendix

Appendix 1. Compiling the UDF for Fluent

The Fluent interface file is a *C* source file contains the heat source data as arrays. Do the following step to compile it.

- Open Fluent program. In Define->User Defined -> Functions -> Compiled, we can open the the dialog as follow
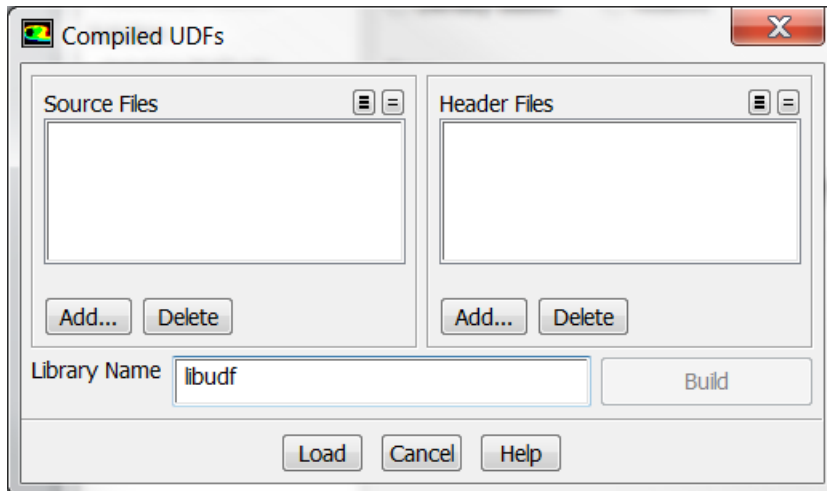
-
- In the *Source files* box, choose Add to add the *C* source file generated by McMeshtran
- Click *Build* to build the source, and the compiled library is placed in the directory where the source code is.
- Click *Load* to load the library.

Appendix 2. Compiling the User Fortran for CFX

The User Fortran code generated by CFX need to be compiled. Following these step:

- Open CFX-pre. Set the working directory to the source code folder.
- In the Tools-> Command Editor, Input the following line
  ! system ("cfx5mkext *test.F*") == 0 or die "cfx5mkext failed";
  - o The *test.F* should be change to the correct directory of the User Fortran file it is located.
  - o