MUFITS

Multiphase Thermal Reservoir Simulator
version 2013.D

Postprocessing Utilities

Author: Andrey Afanasyev

Date: 19 December 2013

Moscow, Russia

Contents

1. Visualization using ParaView **3**

1.1. Introduction 3

1.2. Converting MUFITS output to VTK file formats 4

1.3. Converter options 5

1.4. Using data in ParaView 6

1.5. Visualizing results for Example-H2 6

References **8**

1 Visualization using ParaView

1.1. Introduction

To visualize the results of MUFITS hydrodynamic reservoir simulation in ParaView the MUFITS output files must be converted to a format supported by ParaView [1]. The output can be converted using a special MUFITS module, which is farther called converter. The converter executable names are as follows

- C32.EXE or C64.EXE for 32-bit or 64-bit Windows operating system;
- C32.EXL or C64.EXL for 32-bit or 64-bit Linux operating system;
- C32.EXM or C64.EXM for 32-bit or 64-bit Macintosh operating system.

This executables can be downloaded at the MUFITS reservoir simulator website [2]. The executable should be saved in the subfolder BIN in the MUFITS installation folder. The converter RUN-file must be exactly the same as the RUN-file used for hydrodynamic module. For example, to run the converter on Mas OS the following command must be executed in the terminal

./../../BIN/C64.EXM TITLE.RUN > TITLE.CONV.LOG

Here TITLE.RUN is a hydrodynamic reservoir simulation datafile.

Currently the MUFITS reservoir simulation output can be converted to the following formats supported by Paraview:

- VTK legacy formatted files;
- VTK legacy binary files;
- VTK xml based formatted files;

- VTK xml based binary files (default option).

Currently only parameters defined in the grid blocks are saved in the converted output.

1.2. Converting MUFITS output to VTK file formats

Typically the following steps must be conducted to run a hydrodynamic reservoir simulation and convert the output to VTK formats:

1. Create the RUN-file for the MUFITS hydrodynamic module (Fig. 1.1). We farther assume that this file name is TITLE.RUN

2. Execute the hydrodynamic module (H32.EX* or H64.EX*) using the RUN-file TITLE.RUN to produce the MUFITS hydrodynamic simulation output. The output files are TITLE.MVS, TITLE.GRID.SUM, TITLE.0000.SUM, TITLE.0001.SUM, etc.

3. Execute the converter module (C32.EX* or C64.EX*) using the RUN-file TITLE.RUN. If conversion to VTK xml files is selected than the following files are created in the simulation folder: TITLE.pvd, TITLE.GRID.vtu, TITLE.0000.vtu, TITLE.0001.vtu, etc.). If conversion to VTK legacy files is selected than the following files are created: TITLE.GRID.vtk, TITLE.0000.vtk, TITLE.0001.vtk, etc.)

4. Open ParaView. Select File->Open. Select TITLE.pvd or TITLE.****.vtk files if you want to visualize time series data. If you select TITLE.GRID.vtu or TITLE.GRID.vtk file than you can visualize output saved after section REGIONS.

Recommendations for using the converter are as follows

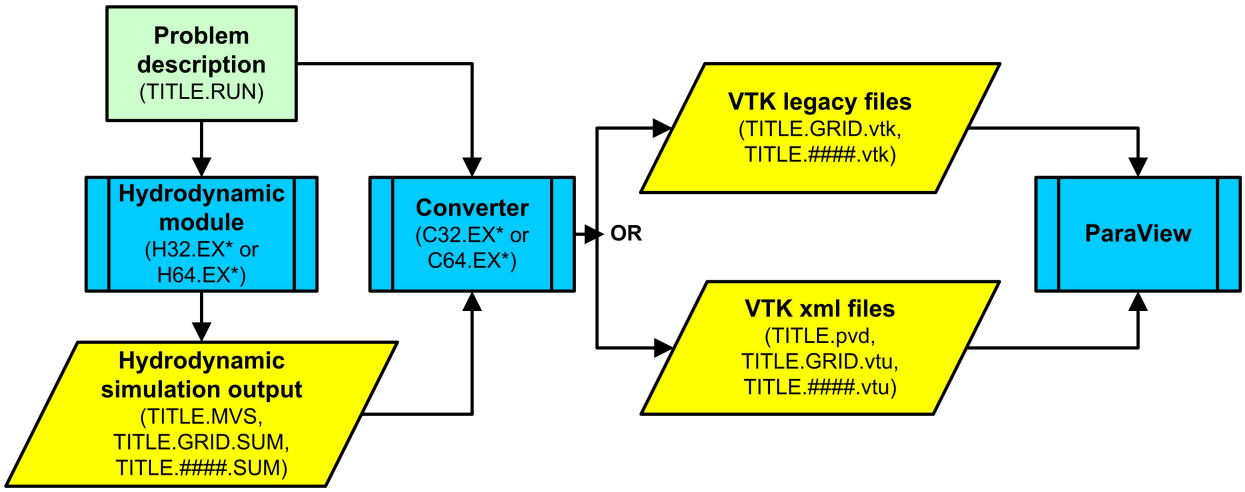

Figure 1.1: MUFITS output files conversion flowchart

- You can run the converter before hydrodynamic simulation is finished. In this case only the available at the moment output files are converted;
- Always specify the keyword SAVEMVS within every MAKE/ENDMAKE brackets in the section GRID to save MVS-files. By the default option these files are not saved. If none MVS-file is found than the conversion is not produced;
- Specify binary MUFITS output (default option) and convert the output files to VTK xml binaries (default option). These options will facilitate a faster conversion and smaller memory requirements.

1.3. Converter options

The converter options can be modified using the keyword CNVPARAM in the section RUNSPEC. This keyword has 7 associated data items, which are as follows

1	-	Must be ASCII if the output of formatted files is requested or BINARY if the output of binary files is requested for the converted data. By the default option the value of this switch is BINARY;

2	-	Must be XML if the output of VTK xml files is requested (default option) or VTK if the output of VTL legacy files is requested ;
3-7	-	The not an arithmetic value for INT1, INT2, INT4, REAL4, REAL8 data type (default 0, 0, 0, NaN, NaN). This value is used for those data items required by VTK formats which are not specified in MUFITS output files.

An example of this keyword specification is as follows

Example of the keyword CNVPARAM

```
CNVPARAM
   1*   VTK    5*0   /
```

1.4. Using data in ParaView

ParaView uses right-hand coordinate system while MUFITS uses left-hand system with z-axis pointing along the gravity acceleration vector. Therefore, by default the MUFITS reservoir simulation output is displayed reflected in ParaView. Use the "Reflect" filter to see the output properly.

1.5. Visualizing results for Example-H2

Let us consider visualization of the porosity field and the CO2 molar fraction field for the Example-H2, which is described at the MUFITS reservoir simulator website [2]. To see the output using Mac OS do the following:

1. Download from the MUFITS website the converter executable C64.EXM and copy it in the subfolder BIN of MUFITS installation;
2. Run the MUFITS hydrodynamic simulation for the Example-H2 as it is outlined in the document Example-H2.pdf;
3. In the terminal navigate to the folder /SIMULATIONS/EXAMPLE-H2/1/;

4. In the terminal execute the command: ./../../BIN/C64.EXM EXAMPLE-H2.RUN > EXAMPLE-H2.CONV.LOG;

5. Open ParaView;

6. Select Edit->Settings->General. Select "Auto Apply" option;

7. Select Files->Open. Open the file /SIMULATIONS/EXAMPLE-H2/1/EXAMPLE-H2.GRID.vtu;

8. In properties window, in the section "Color" select the property PORO. You must see the porosity distribution in the layout. Click on the "Show" button to see the color bar;

9. In the pipeline browser make the object EXAMPLE-H2.GRID.vtu inactive;

10. Select Files->Open. Open the file /SIMULATIONS/EXAMPLE-H2/1/EXAMPLE-H2.pvd;

11. In properties window, in the section "Color" select the property COMP1T. Click on the "Edit" button below. In the color scale editor window input 0.2 as the maximum value. Close the color scale editor window. Click on the "Show" button to see the color bar;

12. Click on the "Play" button just below the center of the menu bar. In the layout you must see the time series data for the CO2 molar fraction distribution in the domain.

References

1. ParaView website. http://www.paraview.org.

2. MUFITS reservoir simulator website. http://www.mufits.imec.msu.ru.