HOW TO SET UP MULTI-ZONE PROBLEMS IN SU2

11.06.2020, i.e. v7.0.5
1. MESH HANDLING

2. CONFIG FILES

3. POST PROCESSING
Disclaimer(s)
Please don’t sue me

- This is by no means my work and there are a lot of people to give credit to ... therefore I just thank the SU2 community 😊

- SU2 is mostly NOT a streamlined developed software => concepts might be inconsistent and bugs occur

- I did my best to get all infos right, but ...
  - there can be errors on the slides and audio track
  - not everything can be mentioned
  - edge cases are not covered

- Who is this for? Everyone ...
  - ... who wants to set up a multizone computation
  - ... and has already some SU2 background

- This presentation is certified 100% equation-free!
What is a Zone?

1. Mesh Handling

- What defines a ‘zone’?
  - Used physics (fluid flow, heat conduction, structural mechanics)
  - Moving mesh (fluid-fluid)
  - Not necessarily geometric connectivity (see image)

- Connection between zones via interface boundaries (optional)

- MULTIZONE single physics and MULTIPHYSICS possible for fluid-fluid cases
1. Mesh Handling

What is a Zone?

- What defines a ‘zone’?
  - Used physics (fluid flow, heat conduction, structural mechanics)
  - Moving mesh (fluid-fluid)
  - Not necessarily geometric connectivity (see image)

- Connection between zones via interface boundaries (optional)

- MULTIZONE single physics and MULTIPHYSICS possible for fluid-fluid cases
1. Mesh Handling
How should I create the mesh?

- Each zone is discretized independently
- Interface Boundary markers have to be ‘geometrical’ pairs
  - KIND_INTERPOLATION= ... specifies how values are communicated over the boundary
- Sidenote: internal structures are separate for each zone (config, solver, etc.)
- MPI: each rank gets a chunk of each zone
  - Important for e.g. Multigrid methods.
1. Mesh Handling
How to create a multizone Mesh

- **MULTIZONE_MESH= NO**
  - Create meshes for each zone just as for single zone cases
  - Mesh has to be specified in each zonal config

- **MULTIZONE_MESH= YES (default)**
  - Easy creation from single zone meshes as shown below
  - Order of the meshes is kept for config files in CONFIG_LIST= (…)

```
NDIM= 2
NELEM= 1234
...
NPOINT= 567
...
NMARK= 3
MARKER_TAG= wall
MARKER_ELEMS= 89
...
```

```
NDIM= 2
NELEM= 9876
...
NPOINT= 5432
...
NMARK= 4
MARKER_TAG= wall
MARKER_ELEMS= 31
...
```

```
NZONE= 2
IZONE= 1
<Mesh_Zone_A.su2>
IZONE= 2
<Mesh_Zone_B.su2>
Multizone_Mesh.su2
```
2. Config Files
Hierarchy

Config files

SOLVER= EULER
ITER= 250
2. Config Files

Hierarchy

default value

SOLVER= MULTIPHYSICS
CONFIG_LIST= (...)
MARKER_ZONE_INTERFACE= (...)

master config

SOLVER= INC_NAVIER_STOKES
MARKER_INLET= (...)
LINEAR_SOLVER= FGMRES

SOLVER= HEAT_EQUATION
MARKER_HEATFLUX= (...)
LINEAR_SOLVER= BCGSTAB

zone A config

zone B config
2. Config Files

Hierarchy

- All config options set in the master are also used in the zonal config (i.e. they are internally copied), except...
- ... config options that are set in the zonal config as well.
- If an option is set in neither of the config files, the default value is taken

Suggestion:
1. Try single zones independently
2. Create master and add zonal configs
3. Transfer common options to master
2. Config Files

Hierarchy

Options to set into the master config:
- SOLVER = MULTIPHYSICS
- MULTIZONE_MESH = YES/NO
- CONFIG_LIST = (zoneA.cfg, ...)
- MARKER_ZONE_INTERFACE = (wallA, wallB, ...)
- TIME/OUTER_ITER = 100

Zonal:
- Do not set interface boundary markers again

Use ‘SU2_CFD <master.cfg>’ to start the simulation
## 2. Config Files

**Example**

<table>
<thead>
<tr>
<th></th>
<th>SOLVER= INC_NAVIER_STOKES</th>
<th>SOLVER= MULTIPHYSICS</th>
<th>SOLVER= HEAT_EQUATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>MESH_FILENAME= Mesh_Zone_A.su2</td>
<td></td>
<td>MESH_FILENAME= Mesh_Zone_B.su2</td>
</tr>
<tr>
<td></td>
<td>MARKER_whatever= (interfaceA)</td>
<td></td>
<td>MARKER_whatever= (interfaceB)</td>
</tr>
<tr>
<td></td>
<td>ITER= 100</td>
<td></td>
<td>ITER= 100</td>
</tr>
<tr>
<td></td>
<td>SCREEN_OUTPUT= (ITER, DRAG)</td>
<td></td>
<td>SCREEN_OUTPUT= (ITER, \ AVG_TEMPERATURE)</td>
</tr>
<tr>
<td></td>
<td>HISTORY_OUTPUT= (ITER, \ AERO_COEFF)</td>
<td></td>
<td>HISTORY_OUTPUT= (ITER, HEAT)</td>
</tr>
<tr>
<td></td>
<td>OUTPUT_FILES= (RESTART,\ PARAVIEW_MULTIBLOCK)</td>
<td></td>
<td>OUTPUT_FILES= (RESTART,\ PARAVIEW_MULTIBLOCK)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>master.cfg</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>zoneA.cfg</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>zoneB.cfg</td>
<td></td>
</tr>
</tbody>
</table>
2. Config Files

Example

- **zoneA.cfg**
  - SOLVER= INC_NAVIER_STOKES
  - MESH_FILENAME= Mesh_Zone_A.su2
  - MARKER_whatever= (interfaceA)
  - ITER= 100
  - SCREEN_OUTPUT= (ITER, DRAG)
  - HISTORY_OUTPUT= (ITER, \ AERO_COEFF)
  - OUTPUT_FILES= (RESTART, \ PARAVIEW_MULTIBLOCK)

- **master.cfg**
  - SOLVER= MULTIPHYSICS
  - MULTIZONE_MESH= YES
  - MESH_FILENAME= multizone.su2
  - MARKER_whatever= (interfaceB)
  - ITER= 100
  - SCREEN_OUTPUT= (ITER, \ AVG_TEMPERATURE)
  - HISTORY_OUTPUT= (ITER, HEAT)
  - OUTPUT_FILES= (RESTART, \ PARAVIEW_MULTIBLOCK)

- **zoneB.cfg**
  - SOLVER= HEAT_EQUATION
  - MESH_FILENAME= Mesh_Zone_B.su2
  - MARKER_whatever= (interfaceB)
2. Config Files

Example

**zoneA.cfg**

```
SOLVER= INC_NAVIER_STOKES
MARKER_whatever= (interfaceA)
ITER= 100
SCREEN_OUTPUT= (ITER, DRAG)
HISTORY_OUTPUT= (ITER, \ AERO_COEFF)
OUTPUT_FILES= (RESTART, \ PARAVIEW_MULTIBLOCK)
```

**master.cfg**

```
SOLVER= MULTIPHYSICS
MULTIZONE_MESH= YES
MESH_FILENAME= multizone.su2
CONFIG_LIST= (zoneA.cfg \ zoneB.cfg)
```

**zoneB.cfg**

```
SOLVER= HEAT_EQUATION
MARKER_whatever= (interfaceB)
ITER= 100
SCREEN_OUTPUT= (ITER, \ AVG_TEMPERATURE)
HISTORY_OUTPUT= (ITER, HEAT)
OUTPUT_FILES= (RESTART, \ PARAVIEW_MULTIBLOCK)
```
2. Config Files

Example

SOVER = INC_NAVIER_STOKES

MARKER_whatever = (interfaceA)

ITER = 100

SCREEN_OUTPUT = (ITER, DRAG)

HISTORY_OUTPUT = (ITER, \ AERO_COEFF)

OUTPUT_FILES = (RESTART, \ PARAVIEW_MULTIBLOCK)

zoneA.cfg

SOVER = MULTIPHYSICS
MULTIZONE_MESH = YES
MESH_FILENAME = multizone.su2
CONFIG_LIST = (zoneA.cfg \ zoneB.cfg)

MARKER_ZONE_INTERFACE = (\ interfaceA, interfaceB)
MARKER_CHT_INTERFACE = (\ interfaceA, interfaceB)

master.cfg

SOVER = HEAT_EQUATION

MARKER_whatever = (interfaceB)

ITER = 100

SCREEN_OUTPUT = (ITER, \ AVG_TEMPERATURE)
HISTORY_OUTPUT = (ITER, HEAT)
OUTPUT_FILES = (RESTART, \ PARAVIEW_MULTIBLOCK)

zoneB.cfg
2. Config Files

Iterations for Dual-Time stepping

- Singlezone steady only uses ITER

ITER= 0..k
2. Config Files

Iterations for Dual-Time stepping

TIME_ITER = 0..n

INNER_ITER = 0..k

- Singlezone steady only uses ITER
- Singlezone unsteady uses TIMER_ITER and INNER_ITER
2. Config Files

Iterations for Dual-Time stepping

- Singlezone steady only uses ITER
- Singlezone unsteady uses TIMER_ITER and INNER_ITER

- In Multizone computations:
  - TIME_ITER -> OUTER_ITER -> INNER_ITER independent of steady-unsteady
  - Exchange of interface data (Coupling) happens with each OUTER_ITER
  - Update in pseudo time with each INNER_ITER
## 2. Config Files

### Iterations for Dual-Time stepping

<table>
<thead>
<tr>
<th>SOLVER</th>
<th>MULTIPHYSICS</th>
<th>HEAT_EQUATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>INC_NAVIER_STOKES</td>
<td>MULTIZONE_MESH= YES</td>
<td></td>
</tr>
<tr>
<td>MESH_FILENAME= multizone.su2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CONFIG_LIST= (zoneA.cfg \ zoneB.cfg)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MARKER_ZONE_INTERFACE= (\ interfaceA, interfaceB)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MARKER_CHT_INTERFACE= (\ interfaceA, interfaceB)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>OUTER_ITER= 100</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**inner_iter= 1**

**zoneA.cfg**

**screen_output= (iter, drag)**

**history_output= (iter, aero_coeff)**

**output_files= (restart, paraview_multiblock)**

**zoneB.cfg**

**screen_output= (iter, avg_temperature)**

**history_output= (iter, heat)**

**output_files= (restart, paraview_multiblock)**

**master.cfg**
### 3. Post- / Midprocessing

**Screen and History output (SU2_CFD –d <config>.cfg)**

<table>
<thead>
<tr>
<th>Solver Configuration</th>
<th>Configuration Details</th>
</tr>
</thead>
</table>
| SOLVER = INC_NAVIER_STOKES | INNER_ITER = 1
SCREEN_OUTPUT = (ITER, DRAG)
HISTORY_OUTPUT = (ITER, AERO_COEFF) |
| SOLVER = MULTIPHYSICS 
MULTIZONE_MESH = YES 
MESH_FILENAME = multizone.su2 
CONFIG_LIST = (zoneA.cfg \ zoneB.cfg) | OUTER_ITER = 100
SCREEN_OUTPUT = (OUTER_ITER, DRAG[0], AVG_TEMPERATURE[1])
HISTORY_OUTPUT = (ITER, AERO_COEFF[0], HEAT[1])
OUTPUT_FILES = (RESTART, PARAVIEW_MULTIBLOCK) |
| SOLVER = HEAT_EQUATION | INNER_ITER = 1
SCREEN_OUTPUT = (ITER, AVG_TEMPERATURE)
HISTORY_OUTPUT = (ITER, HEAT) |

---

*zoneA.cfg*

*master.cfg*

*zoneB.cfg*
3. Postprocessing

Output Files

- **Option A:** Write zonal output independently and load consecutively into your preferred tool (Paraview, Tecplot, etc.) ...

- **Option B:** ... or use OUTPUT_FILES= (PARAVIEW_MULTIBLOCK). Load that into Paraview via the .vtm file

- **Sidenote:** Default name for vtm-folder and history is the master-config filename
3. Postprocessing

Paraview Multiblock (.vtm)
3. Postprocessing
Paraview Multiblock (.vtm) animation
Discrete Adjoint Gradients

- Gradients via discrete adjoint method are available just like in single zone simulations.

- Explicitly set OBJECTIVE_FUNCTION and OBJECTIVE_WEIGHT in each zonal config, even for no zonal contribution to the objective function.

- Python scripts, e.g. shape_optimization.py, do not work with in multizone in the moment.

Image courtesy: Ole Burghardt

- Image: Heatflux sensitivities of the inner pin

https://su2code.github.io/tutorials/Inc_Heated_Cylinders/
Finishing Remarks

“I slept the last 30 minutes... what did I miss?” – Random Listener

We covered (in a basic manner):

- Mesh creation
- Basic structure of the multizone config files
- Output considerations
Finishing Remarks

“I slept the last 30 minutes... what did I miss?” – Random Listener

We covered (in a basic manner):

- Mesh creation
- Basic structure of the multizone config files
- Output considerations

... so now everything should be easy, right?
Finishing Remarks

“I slept the last 30 minutes... what did I miss?” – Random Listener

We covered (in a basic manner):

- Mesh creation
- Basic structure of the multizone config files
- Output considerations

... so now everything should be easy, right?

If there are problems:

- See website https://su2code.github.io for more information ...
- ... if that does not help take a look into ‘TestCases’ folder (e.g. ‘grep –r MULTIPHYSICS’)...
- ... or ask a question https://www.cfd-online.com/Forums/su2/ and bug reports https://github.com/su2code/su2/issues include me via @TobiKattmann

Thank you for tuning in and I hope you enjoy working with SU2 😊