Building and meshing computational domains for industrial CFD with SALOME and *Code_Saturne*

Nicolas Tonello
Director

Renuda, 329-339 Putney Bridge Road, London, SW15 2PG, UK
For further information, please contact:

Nicolas Tonello
Director
RENUDA

nicolas.tonello@renuda.com

T: +44 (0)20 3371 1709
1. Renuda at a glance
2. Complex CFD domains creation
3. Conclusions
1. Renuda at a glance
• CFD Specialists
  • Consulting, Software development, Training
  • Fully independent
  • UK, France, Germany

• Blue Chip Clients
  • Applications from single phase pipe flow to turbomachinery, multiphase flow, coupled heat transfer, reactive flow, mechanical calculations
  • Industries: transport, automotive, processing, nuclear, power generation, civil engineering

• Reputation built on
  • Skills
  • Efficient solutions to difficult problems
Software and Hardware Tools

• Commercial software:
  ▪ Siemens – STAR-CCM+
  ▪ ANSYS

• Open source software chains
  ▪ CAD: SALOME
  ▪ Mesh: SALOME and snappyHexMesh
  ▪ CFD: *Code_Saturne* and OpenFOAM
  ▪ Analysis: Paraview

• Mix of local and remote computing
  ▪ Local multi-core PCs
  ▪ Access to HPC on hundreds of compute cores at the Hartree Centre, UK
Research Partnership And Collaborations

- Research and development is very important
- Collaborative research relationship with EDF R&D on the development of Code_Saturne
- Collaboration with the SALOME teams:
  - Development of GUI for specialised steam turbine code
    - From CAD to Analysis
  - Beta testers for the parametric design module SHAPER
- Part of the UK Consortium on Turbulent Reactive Flow
- Collaboration with different universities and research labs
  - University of Manchester
  - Daresbury Laboratory (Science and Technology Facilities Council) – HPC research and application
  - University of Edinburgh (software parallelisation)
2. Complex CFD Domain Creation
CFD Model Creation

• CFD starts with the creation of a computational domain – a region in space which defines the bounds of the problem of interest and on whose boundaries, boundary conditions can be specified
• Creating the domain may be done from scratch or by importing an already existing CAD file – which may then also be substantially modified
• Once the domain has been created, for CFD it must be surface and volume meshed
• Finally, surfaces must be defined on which boundary conditions may be applied to define the operation of the system of interest
Automating

- Complexity can present itself under different forms:
  - Shape sizes
  - Domain size
  - Shape complexity, intersection of parts
  - Number of parts
  - Etc.
- When dealing with systems which contain a large number of repeated parts and large dimensions, and to parametrise the domain creation to provide flexibility downstream in CFD study, a form of automation is highly advantageous.
- We present an illustration of CAD and mesh creation using the combined use of the powerful scripting possibilities of SALOME and Code_Saturne, with the example of a parametric model of PWR nuclear fuel assemblies.
PWR Nuclear Fuel Assemblies

Credit: ASN
https://www.asn.fr/Lexique/Assemblage-combustible

Credit: World Nuclear Association
Les interactions entre assemblages déformés conduisent à des endommagements de grilles en manutention.
CAD and Meshing Challenge

- Representing a series of fuel assemblies without and with deformation. Additional challenge: deformation so that a grid assembly comes into contact with the side.

- Very long, thin domains, o(a few meters in height)

- 8 grilles

How to create the domains in an efficient and flexible manner?
Principles

• Reduce the overall system to the smallest possible elements, using the fact that they are repeated as well as symmetries
• These elements become the building blocks, which can then be assembled together in different configurations:
  ▪ Domain size
  ▪ Deformation or no deformation
• Decompose the process into different stages
• Script every stage of the process
  ▪ Use the automatic script dumps from Salome
  ▪ Revise the coding to include clear variables and parameters names
• Run the different scripts in succession to create the domain and the mesh
• Here, the complete assembly is done only in the meshing stage
• All the operations are carried out in SALOME
• The meshes are created with Netgen and Distene’s MGTetra
• The meshes are extruded and wall layers are created in the final meshes using Code_Saturne
Sub-Element

• Volume sub-element
Volume sub-decomposition

- The main brick is further decomposed in elements which can be used later to represent the different configurations desired
Surface sub-decomposition

- Likewise, surface must be defined for the domain

outlet  walls Tubes droits  walls Grilles droits

walls Y

cells Ext

cells Ext

inlet
Example of Initial Reduced Model

- Meshing blocks are created, which can then be assembled in domains of any size directly in SMESH
Utilising $Code\_Saturne$ for Meshing

- $Code\_Saturne$ is applied to the complete mesh in order to create both
  - The wall layers
  - The extrusions to represent the inlets and outlets
- The entire meshing process is parametrised
Full Case without Deformation

- Illustration of the CAD model
Full Case with Deformation

- Deformed CAD. The amount of deformation is an input parameter of the script.
Full Case with Contact

• In the more extreme case, the grid assembly is in contact with the side wall of the domain
Deformation Scenarios

- Zoom on the top part to show the displacement and contact

No deformation  Reduced gap  Full contact
Final Mesh

- Illustration of the final, extruded mesh for the full-contact configuration

Outlet_Z (outlet for ¼ of the domain)

Inlet_1  Inlet_2  Inlet_3
Process validation: CFD

- Verification over a few tens of iterations with *Code_Saturne*
3. Conclusions and Future Work
Conclusions and Future Work

- **SALOME and Code_Saturne** form a very powerful combination, which Renuda is applying in very different industrial settings and purposes. Renuda is also participating in its development.
- The SALOME set of tools gives a lot of flexibility to prepare and modify models and to mesh them.
- Further progress can be made on usability and speed.
- Wall layers, which are at the heart of CFD simulations, remain a difficult topic for meshers and significant progress is being made with EDF to implement solutions.