Meshing and Post-processing Capabilities of Open Cascade SALOME and its Compatibility with Solver Code Saturne for setting up physics and run the simulation

[1]Manoj Kumar AP, [1]Suresh KV, [1]Yogish Rao
[1] Alvas Institute of Engineering and Technology, Moodbidri, Karnataka, India

Abstract: This paper describes about the features of open cascade SALOME for meshing engineering components, post processing using Solver Code Saturne to run the simulation and post processing by ParaVis to analyze the results. The objective of this study is to explore the features of SALOME/Saturne as an alternate to commercially available CFD tools without compromising the quality of the result. The study includes the comparison of SALOME/Saturne with other commercial tools with respect to cost, ease of carrying out preprocessing and post processing and end result validation. Modeling is also possible with this tool and user can model any part in the Geometry module covered in this paper. In the current paper, an exhaust manifold is taken for carrying out simulation to determine pressure drop and velocity inside the system. The step by step procedure for meshing is well explained and 3D images of pressure drop and velocity are shown. In another example, a simple pipe flow is taken as example and simulation has been carried out using SALOME/Saturne to determine the pressure drop and mass flow rate calculations. The results are validated through theoretical calculation.

1. Introduction

The point of view of this study is on practical side, i.e. usability, software ergonomics and ability to produce high quality mesh are emphasized instead of meshing algorithms. In selecting the software open source has not been a necessary condition since also some proprietary mesh generators are often used with open source solvers. However, a rather inexpensive license pricing is desirable since typically the users of open source software have limited monetary resources. Mesh generation is an essential part of the solution procedure which often consumes the most of the human resources. The accuracy and efficiency of computation depends upon element or cell shape and size, which in turn depends upon speed and numerical efficiency.

For CVM and in most cases FEM, hexahedron elements are more favorable to numerical efficiency, but there isn't any general automatic and robust mesh generation algorithm available for such a mesh type. Due to increase in computational power, the number of elements doesn't have that much significance any more. This has lead to use of automated tetrahedral mesh generators both in CVM and FEM. These meshing tools can produce relatively high quality mesh in just tens of seconds or couple of minutes. The mesh quality is often good enough for structural analysis but in CFD results are more sensitive on mesh type and quality and the use of tetrahedral mesh can lead to a high number of cells to achieve the same computational accuracy than when using hexahedral cells. One solution for the element or cell type issue is the use of a hybrid mesh, i.e. a mesh containing both hexahedral and tetrahedral elements or cells. This allows e.g. creation of hexahedral mesh is critical areas manually and mesh the rest of the volume using automatic mesh generation and tetrahedral elements or cells. There are also methods to automatically create hybrid mesh in arbitrary geometry. This area is not covered in this document.

Code_Saturne solves the Navier-Stokes equations for 2D, 2D-axisymmetric and 3D flows, steady or unsteady, laminar or turbulent, incompressible or weakly dilatable, isothermal or not, with scalars transport if required. Several turbulence models are available, from Reynolds-Averaged models to Large-Eddy Simulation models. In addition, a number of specific physical models are also available as "modules": gas, coal and heavy-fuel oil combustion, semi-transparent radiative transfer, particle-tracking with Lagrangian modeling, Joule effect, electrics arcs, weakly compressible flows, atmospheric flows, rotor/stator interaction for hydraulic machines. ParaVis is a post processing application available in SALOME platform, where results can be viewed and analyzed in 3D.

There are many commercially available software's available to carry out various types of analysis as shown in Figure 1. The license cost is the primary concern, where any organization needs to take a critical decision to buy it, which in turn depends upon budget allocation for software procurement. If the client is not ready to bear this expense, organization has to find an alternative ways to meet the current software requirement, which leads to opting freeware software (open cascade platform), where one can carry out similar analysis without compromising the quality of results. The following flow chart gives an idea of commercially available software and open cascade tools.

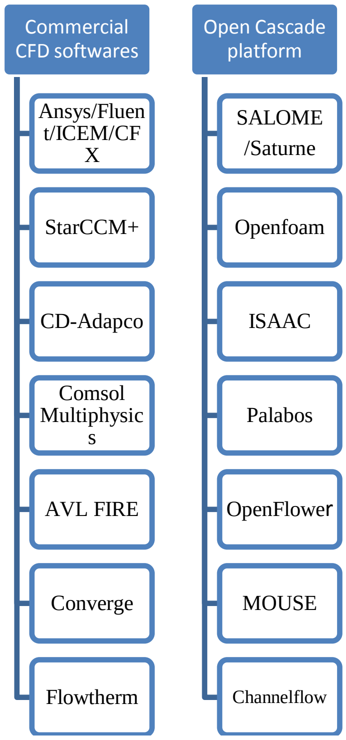


Fig 1 List of Commercial and Open Platform CFD Tools

There are many open-cascade CFD tools are available, including Open foam, Salome/Saturne, ISAAC, Palabos, Open flower etc, where the user can carry out variety of simulation, without lending money on purchasing license. In the current paper SALOME/Saturne tool is taken into consideration and the ease of carrying out modeling, preprocessing, case set up/simulation and post processing is well explained with few examples.

In the first example, an exhaust manifold with 3 inlets and single exhaust is analyzed to determine the pressure drop and velocity in the entire system. In another example, a simple pipe flow is taken as example and simulation has been carried out using SALOME/Saturne to determine the pressure drop and mass flow rate calculations. The results are validated through theoretical calculation.

1.1 Salome

SALOME is a general graphical environment for numerical computing using spatially discretized, mesh-based methods like finite element or control volume method. The environment contains separate working modes for geometry creation or manipulation (Geometry), meshing (Mesh), solver management (Supervisor), post-processor (Post-Pro), and communication module (MED). The software is open source software under GNU Lesser General Public License (LGPL). The geometry management in the environment is based on Open Cascade geometry kernel and Visualization Toolkit (VTK) visualization library. The software is available as binary form for several Linux distributions and in source code. SALOME is also available in windows version, where the user can use GUI's for modeling and meshing purpose.

SALOME supports via Open Cascade the following solid geometry file formats: ACIS, SALOME native BREP, IGES, and STEP. Solid geometry can be exported in ACIS, BREP, IGES, and STEP formats. Also STL (both ASCII and binary) can be used as the model export format. Created mesh can be exported from SALOME in DAT, MED, I-deas UNV, and STL format (ASCII or binary). I-deas UNV format is an ASCII format that has an open format specification. It is design especially for FEM data and is suitable for all mesh data storage. There is a utility for data conversion from UNV format to Open FOAM internal mesh representation. The STL format is used for surface mesh export and is not usable for CFD mesh export. At the moment only mesh in UNV format can be used with Open FOAM.

SALOME has quite extensive capabilities in creation and manipulation of solid geometry. New geometries can be created either using CSG functionality or BREP operations. New solids can be constructed from manipulated existing surface components by adding new surface so that a closed volume is produced. SALOME also has good shape healing operations too e.g. find and remove small edge and surface components and to combine surface components.

1.2 Modeling Capabilities

Salome Geometry module provides the toolset allowing creating a vast range of geometrical objects, from points to complex extrusions.

- Create Basic objects - points, lines, circles...
- Create Primitives - cubes, spheres, cones...
- Create Complex objects by extrusion, rotation, interpolation of other objects.
- Create and edit Groups of objects of lower dimension, which belong to the objects of higher dimension.
- Build by blocks faces from edges and solids from faces.
- Explode objects of higher dimension into sub-objects of lower dimension.
- Create Topological objects - edges, wires, shells...

SALOME got similar features to other modeling tools like CATIA, Pro/E, Solid Works etc. It's simple options make user to learn the modeling quickly. In Geometry module you can import and export geometrical objects from/into BREP, IGES, STEP, STL, XAO, VTK (only export) and others files. The mechanisms of import and export are also implemented via plugins, which gives the opportunity to expand the range of available formats by adding more plugins (for example, CATIA 5 or ACIS). If a plugin supports import of materials associated with shapes, these shapes are grouped corresponding to the imported materials. For the moment STEP import is the only plugin that supports this feature. It is one of the unique features of SALOME, which made its application wider in various companies. It also allows carrying out various transformations and repairing operations with the given geometry. Some tools in GEOM allow creating shapes, basing the design on imported pictures (engineering drawings, nautical charts etc. This is another feature which distinguished SALOME from other modeling tool. It also handles wrapping function to deform the surface. The GEOM python package essentially contains, python interface **geomBuilder.py** to import/export, create and transform geometrical objects, manage fields, use measurement tools. The utility functions within Python module **geomtools.py** to handle GEOM items in SALOME study includes,

- Add or remove a shape;
- Display or erase a shape in the viewer;
- Completely delete a shape (undisplay, unpublish, and destroy it);
- Manage the selection in the object browser.

1.3 Meshing Capabilities

SALOME is capable of producing tetrahedral, hexahedral and prism meshes. Hexahedral mesh generation requires hierarchical modeling from volume edges and volume faces to mesh blocks. This can be done also for CAD geometries but is easier for geometries that are created in SALOME. In this paper hexahedral mesh generation is not presented. Prism meshes are generated by extruding an existing mesh surface to a volume. This method can be useful for generating a boundary layer mesh for CFD.

In SALOME there are quite good control over the mesh topology in general. The mesh can be composed from sub-meshes which may simplify the meshing and allows larger number of cells to be produced; this is because the meshing and optimisation algorithms don't have to manage the whole set of cells at once. If the generated mesh contains some local anomalies, the user can manually fix this even in cell vertex level. This means the user can define individual cells by defining corner points (vertices), edges (lines connecting vertices) and finally cells. Automatic mesh generation using tetrahedron elements/cells is relatively fast with both basic methods (2D surface meshing with either Netgen or Mefisto); the 3D meshing is done using Netgen meshing routine.

2.0 3D Meshing Procedure

The geometry (Exhaust manifold) is imported into SALOME workflow manager as shown in Figure 2 (a). The files in the form of BREP, STEP, IGES, STL and XAO can be successfully imported into SALOME. In the current example, a STEP file of exhaust manifold has been imported.

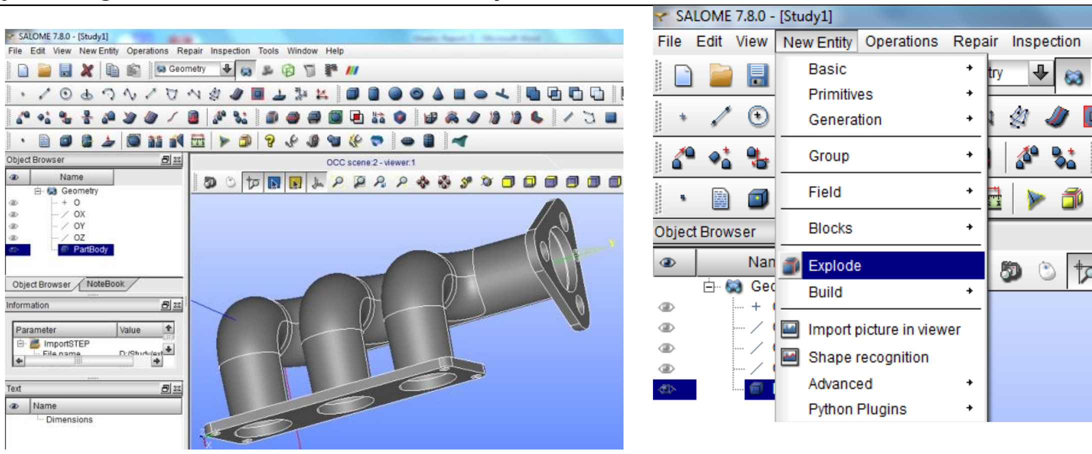


Fig 2(a) Imported geometry in SALOME geometry module (b) Explode option in SALOME

Once the geometry is imported, next step is to trim out unnecessary faces, and to retain only flow domain. The Explode option is used to segregate all the faces associated with the solid geometry as shown in Figure 2(b). Once the faces are exploded, delete the unnecessary faces from it.

The final flow domain (after removing unnecessary faces) is shown in Figure 3 (a).

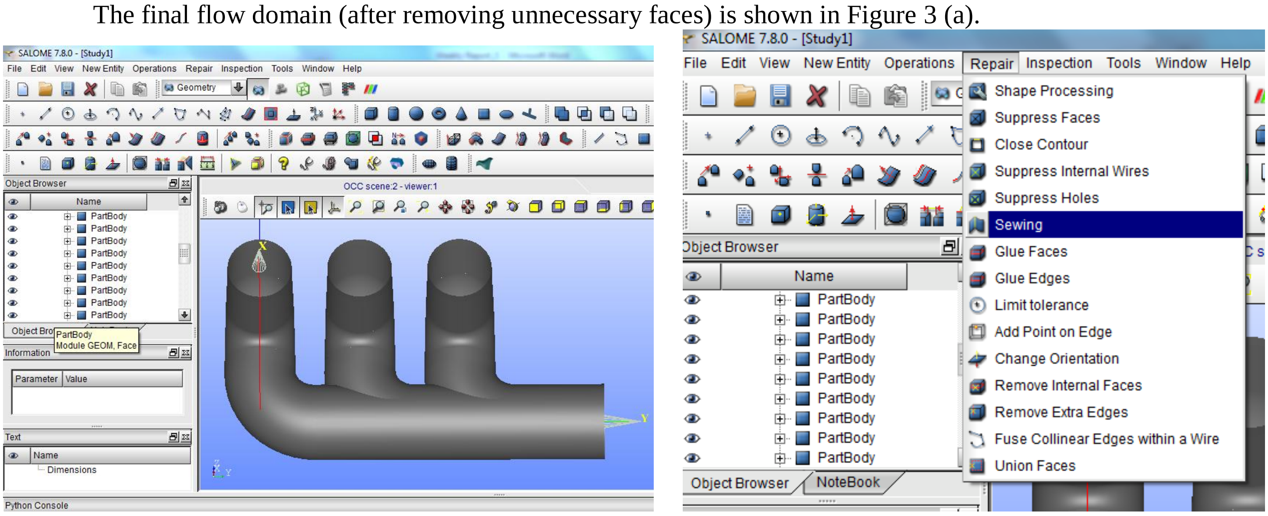


Fig 3 (a) Final Flow domain after explode option (b) Sewing Operation in SALOME

Sewing operation allows uniting several faces (possibly contained in a shell, solid or compound) into one shell as shown in Figure 3 (b). Geometrically coincident (within a specified tolerance) edges (or parts of edges) of different faces are replaced by one edge thus producing a shell of faces with shared boundaries. It is necessary to get closed boundaries, in order to achieve meshing. Hence, open boundaries should be closed with faces. This operation is done by Suppress holes option as shown in Figure 4 (a).

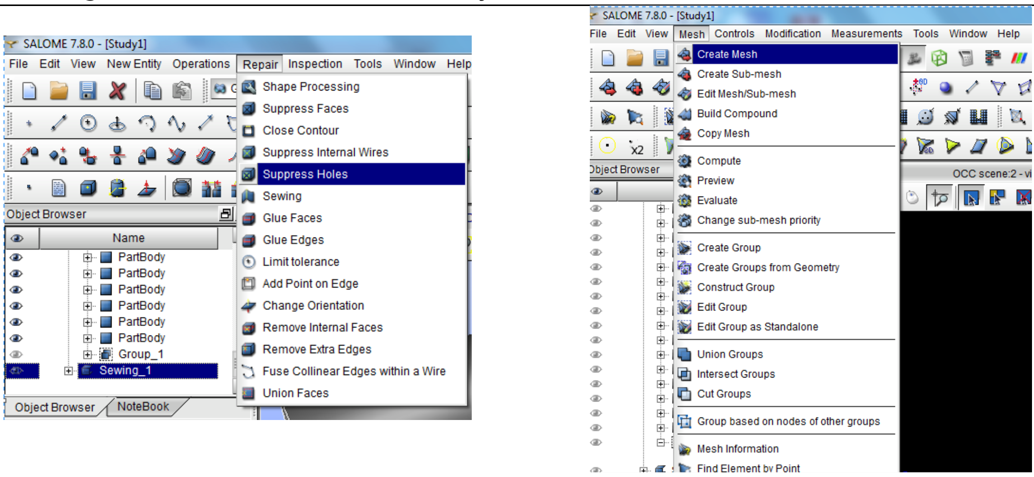


Fig 4(a) Suppress holes option in SALOME (b) Mesh creation option in SALOME

Open Mesh module and click on create mesh as shown in Figure 4 (b). In geometry option select Solid_1 and in 3D option select Tetrahedron (Netgen) algorithm and in hypothesis select Netgen 3D parameter. In Netgen 3D parameters, select the required size and fineness. In Add hypothesis select Viscous layer and fill all the required data as shown in Figure 5 (a).

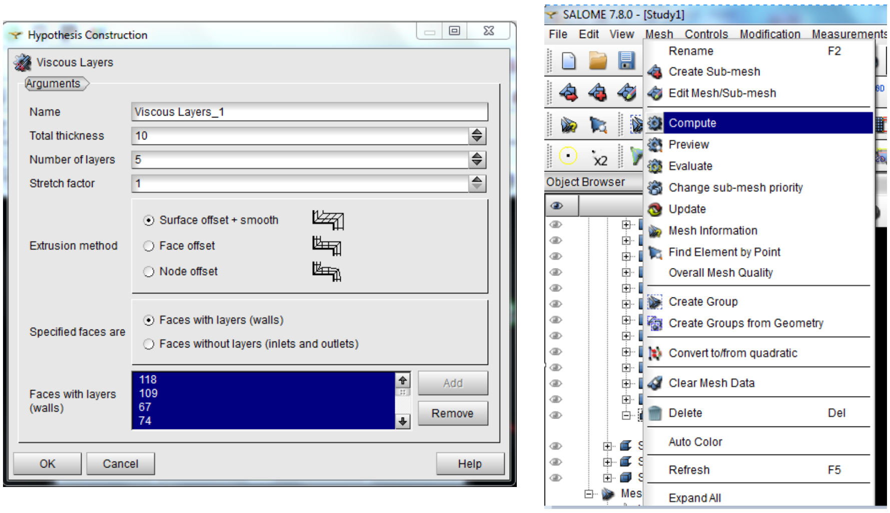


Fig 5 (a) Viscous layer option in SALOME (b) Compute mesh option

Now click on 2D, and select Netgen 2D. In hypothesis, select Netgen 2D parameters and click on it. Enter the required values in the tab shown below. Now click on 1D option as shown below and in algorithm select Wire discretisation and in hypothesis, click on automatic length option. Next step is to right click on Mesh in the tree and click on compute as shown in Figure 5(b). The final mesh is shown in Figure 6.

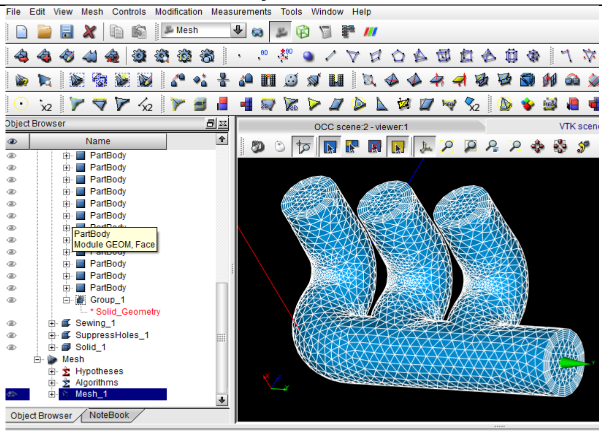


Fig 6 Final Meshed model

2.1 Creating Cut Planes

The cut-planes help to view the mesh qualities inside the solid mesh. We can create plane at required location and check whether the mesh is proper or not. This property is very much helpful in analyzing mesh quality in sensitive regions, where you need to find some values say pressure, temp or mass flow rate etc. Right click on mesh and select clipping and click on New-Relative. Set distance as 0.5, a plane can be seen on the mesh exactly at the middle location. Now click on apply. The mesh-cut view is shown in Figure 7 (b).

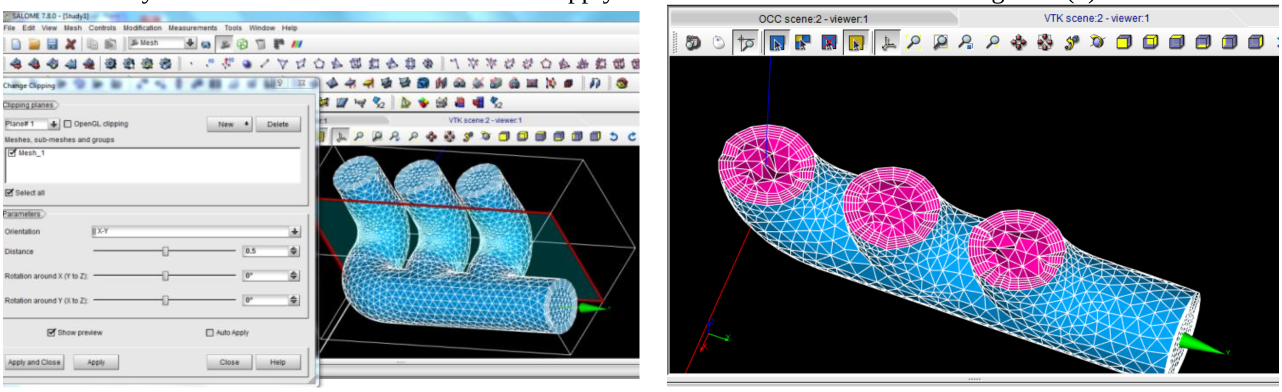


Fig 7(a) Plane used to cut the meshed model (b) The cut view of meshed model

3.0 Case Set Up Using Solver Code Saturne

This section will explain, how case set up can be build in Solver Code Saturne. Once the mesh is generated from SALOME, it can be exported in the form of .MED file. It can be successfully imported into solver code saturne. Once the mesh is imported, one has to select type of module. There is variety of modules available including steady/unsteady flow algorithm, multiphase treatment, atmospheric flow, gas combustion, compressible model, pulverized fuel combustion, ground water flows, electrical models etc. In the current example, a steady state flow model has been selected as shown in Figure 8.

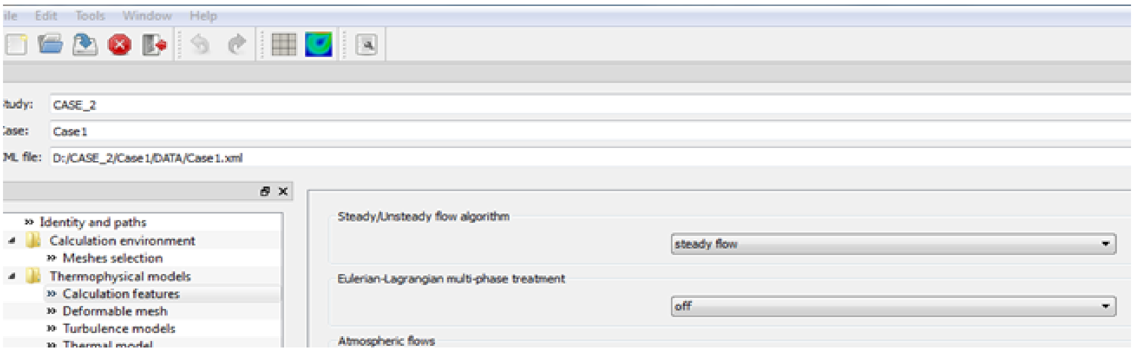


Fig 8 Selection of steady flow simulation in solver code Saturne

In the turbulence model, simple laminar flow model is selected. In fluid properties, water is taken as fluid with density 1000 kg/m3, viscosity 1 Pa.s and specific heat as 4800 J/Kg/K.

The following boundary conditions have been taken for the current model. 1. Inlet 2. Outlet 3. Wall, as shown in Figure 9.

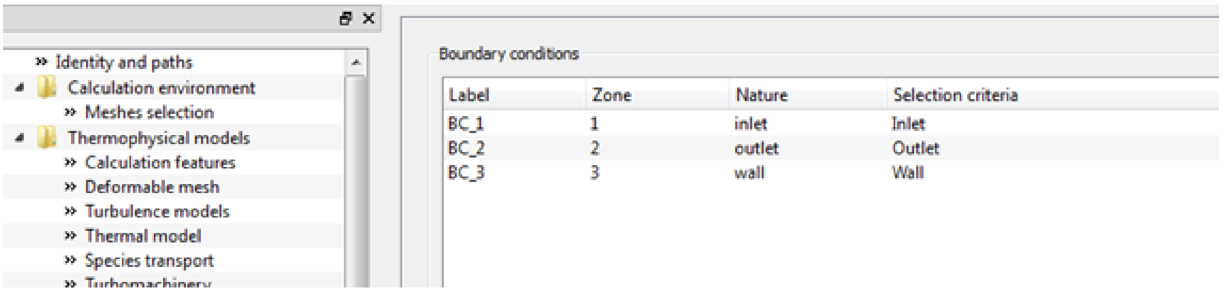


Fig 9 Boundary conditions chosen in Saturne

In Numerical parameter, the user can keep Global and Equation parameters as constant. In Pseudo time step option, we can adjust the number of iteration based on application and convergence criteria. In the current example number of iterations is taken as 100. In output control, one has to select the mesh and results must be written in each time step.

In prepare batch calculation option; standard run type has to be selected as shown in Figure 10. The file must be saved before given it for run.

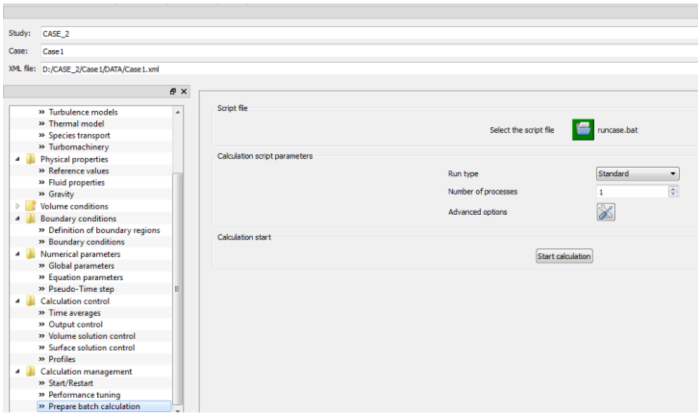


Fig 10 Preparation of batch calculation in Saturne

3.1 Advantages of SALOME/Saturne Preprocessing

It is easy to carry out meshing in SALOME, where any complex geometry can be meshed without much effort and very quickly. There is a wide variety of algorithms available in SALOME to carry out meshing process including tetrahedral, Hexa, MG-Hexa, MG-Tetra etc. One of the important features in SALOME is the use of Netgen 1D-2D-3D algorithm, which will carry out the meshing within few minutes, taking into consideration of bends, cracks, corners etc. It will provide extra refinement at the critical regions, which will increase the mesh quality and in turn results. Any complex geometry can be meshed using this algorithm, where the user needs to just click on this algorithm. It makes SALOME more popular, when comparing with other commercial CFD tools. Netgen 1D-2D-3D algorithm meshes 1D, 2D and 3D elements together. The meshing procedure in commercial CFD packages like StarCCM+, AVL FIRE, CD-Adapco, Converge etc is a tedious job, where the user need to follow so many procedures to obtain high quality mesh. The meshing time is also more when compared with SALOME.

Case set up is also easier in SALOME, when compared with other CFD tools. SALOME is compatible with solvers like CAE linux, Saturne, Aster, Elmer etc. where the user can enter/choose the type of simulation, boundary conditions, and properties of fluid, equation control, turbulent and other related models and simulation control.

4.0 Post Processing using ParaVis

ParaVis is a module coming along with SALOME, where the user can perform post processing activities. This module provides a wide range of applications to view and analyze the results in a detailed manner. Although it looks little harder in initial use, after working few examples will make user feel comfortable. The pressure variation inside the exhaust manifold is shown in Figure 11.

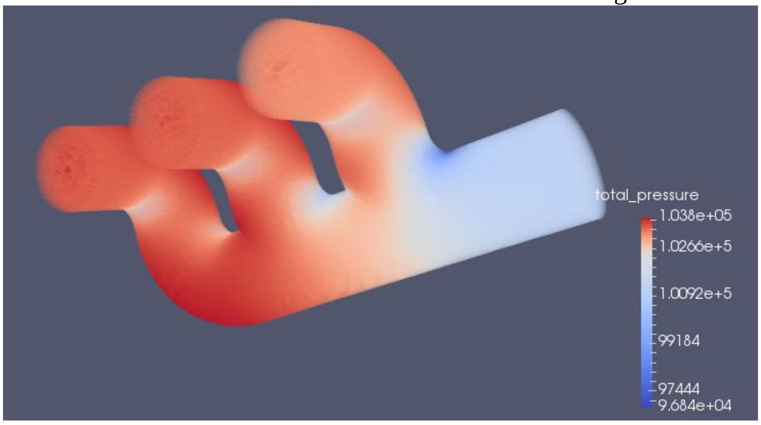


Fig 11 Pressure distribution inside the exhaust manifold

Figure 11 illustrates the pressure drop from inlet to outlet. The range of pressure have been showing decreasing trend from first limb to second, third and finally to outlet. There is a pressure drop of 7KPa is observed from inlet to outlet.

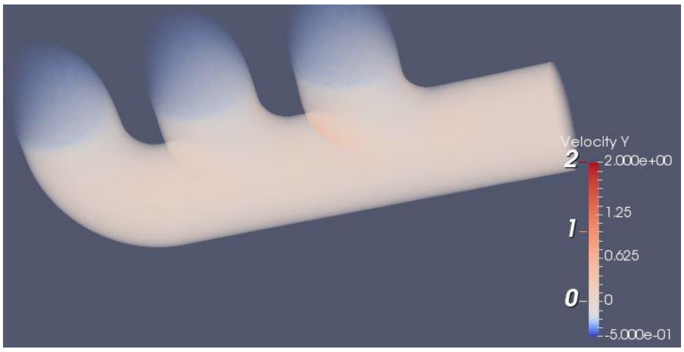


Fig 12 Velocity distribution inside the exhaust manifold

There is a slight increment of velocity from inlet to outlet as shown in Figure 12. The velocity at outlet at outlet is the summation of velocities at Limb1, 2 and 3 respectively. The increase in velocity is due to addition of kinetic energies exerted from each limb and also decrease in the cross section of limbs at a point where it meets common pipe.

5.0 Hardware Requirements

Compared to other commercial CFD softwares, SALOME requires minimum system requirements, so there will not be huge investment for server/processor/RAM. SALOME platform execution requires minimum of 512Mb RAM (1024Mb recommended) and at least 500Mb of swap space. SALOME full installation requires ~ 2 GB of disk space, but during the installation (using SALOME Installation Wizard) it might need up to 2 GB of additional free disk space, for storing of temporary files. Intel (R) Core (TM) i5-6500T CPU@2.50 GHz processor is enough, which will process the data with optimum speed.

6.0 Validation of Pipe Flow Analysis using SALOME /Solver-Code Saturne with Theoretical calculation

This section explains the simulation of simple cylinder, where laminar flow model is considered. The results are also validated with theoretical calculation. It was observed that, simulation and theoretical results are in well agreement with each other. The meshed geometry and 3D image of pressure distribution are shown in Figure 13 and Figure 14 respectively.

(i). Meshed Geometry

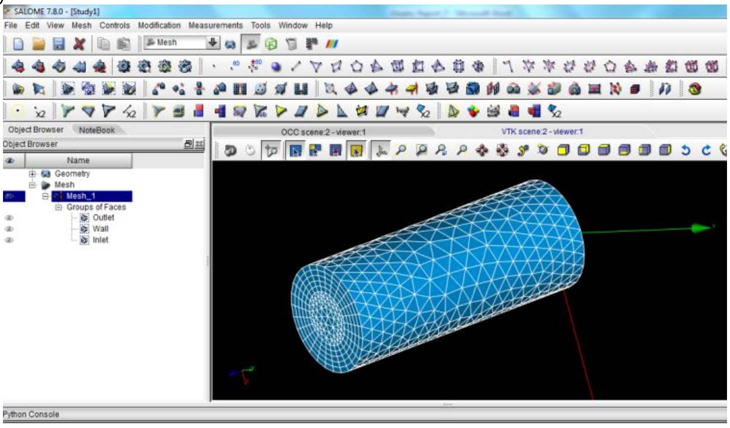


Fig 13 Meshed Cylindrical Model

Cylinder Dimensions:

Diameter: 0.4 m
Height: 1 m

Boundary Conditions:

Inlet: Velocity- 1 m/s
Outlet: 1 bar
Wall: Smooth Wall Condition
Turbulence Model: Smooth Laminar flow
Type of flow: Steady

Fluid Properties: Density- 1000 kg/m3
Viscosity- 1Pa.s
Specific Heat- 4800 J/kg/K

(ii). Post Processing Using ParaVis

Pressure Distribution:

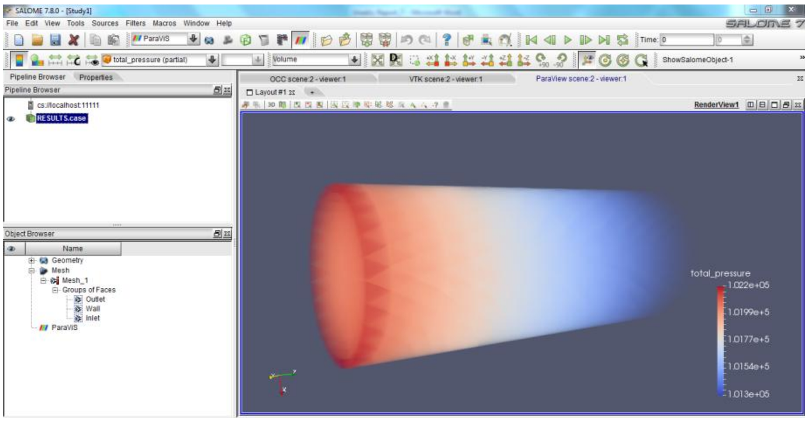


Fig 14 Pressure distribution inside the cylinder

Pressure Drop from the Plot:

Maximum Pressure: 1.022 E5 Pa
Minimum Pressure: 1.013 E5 Pa

Difference: 1.022E5 – 1.013E5 = 0.009 E5 Pa = 900 Pa
Pressure Drop = **900 Pa**

(iii). Theoretical Calculation of Pressure Drop:

The first step is to determine whether the flow is laminar or turbulent

$$\mathrm{Re} = \frac{\rho v D}{\mu}$$

$$\mathrm{Re} = \frac{1000 * 1 * 0.4}{1} = 400$$

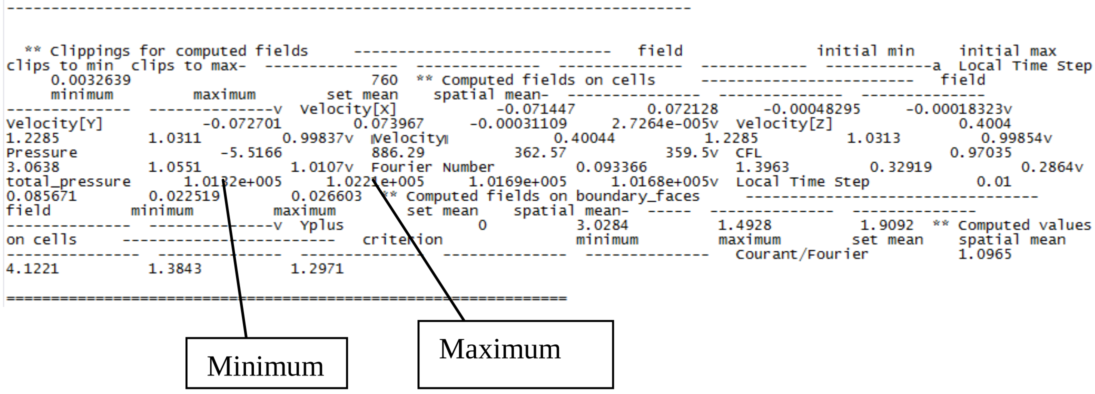
Since 400 < 2000, the flow is Laminar.
Therefore, Neglecting friction, the pressure drop can be calculated from following formula

$$\Delta P = \frac{H}{D} * \frac{\rho}{2} * \varpi^2$$

Where, H is height of cylinder, D – Diameter, ρ is density of fluid, ϖ is the average velocity
The average velocity from the simulation result is 1.028 m/s.

$$\Delta P = \frac{1}{0.6} * \frac{1000}{2} * 1.028^2$$

ΔP = 884 Pa
Therefore,
Pressure Drop
Simulation: 900 Pa
Theoretical: 884 Pa $\cong 900$ Pa
The Pressure Output from simulation is given below.



```
-----------------------------------------------------------------------------


  ** Clippings for computed fields     -----------------------------   field              initial min     initial max
clips to min  clips to max-  ---------------  --------------  --------------  ------------  ------------a  Local Time Step
     0.0032639                            760  ** Computed fields on cells      -------------------------   field
     minimum         maximum          set mean    spatial mean-  ---------------  ---------------  ---------------
--------------  --------------v  Velocity[X]          -0.071447         0.072128     -0.00048295     -0.00018323v
Velocity[Y]            -0.072701         0.073967     -0.00031109     2.7264e-005v  Velocity[Z]             0.4004
1.2285          1.0311          0.99837v  Velocity          0.40044           1.2285          1.0313           0.99854v
Pressure                -5.5166           886.29          362.57          359.5v  CFL                     0.97035
3.0638          1.0551           1.0107v  Fourier Number            0.093366         1.3963          0.32919          0.2864v
total_pressure       1.0132e+005      1.0221e+005      1.0169e+005      1.0168e+005v  Local Time Step          0.01
0.085671        0.022519         0.026603  ** Computed fields on boundary_faces     ---------------------------------
field           minimum          maximum           set mean    spatial mean-  -----  --------------  --------------
--------------  --------------v  Yplus                   0            3.0284          1.4928          1.9092  ** Computed values
on cells     ------------------------   criterion               minimum         maximum          set mean    spatial mean
---------------  ---------------  ---------------  --------------  --------------  Courant/Fourier           1.0965
4.1221          1.3843           1.2971


=============================================================
```


Mass flow rate Calculation (theoretical)
Theoretical

$$\dot{m} = \rho * Q$$

Where ρ = 1000 kg/m^3

Discharge Q = Area * Velocity

$$Q = \frac{\pi * D^2}{4} * \text{Velocity}$$

$$Q = \frac{\pi * 0.4^2}{4} * 1$$

Q = 0.125 m^3/s

$$\dot{m} = 1000 * 0.125$$

$\dot{m}$ = 125 kg/s

The simulation output of mass flow rate is given below

```
 ** BOUNDARY MASS FLOW INFORMATION
    -----------------------------

--------------------------------------------------------------
Boundary type          Code     Nb faces           Mass flow
--------------------------------------------------------------
Inlet                    2         252        -0.124532570E+03
Smooth wall              5         936         0.000000000E+00
Rough wall               6           0         0.000000000E+00
Symmetry                 4           0         0.000000000E+00
Free outlet              3         268         0.124532570E+03
Free inlet              14           0         0.000000000E+00
Convective inlet        16           0         0.000000000E+00
Free surface            15           0         0.000000000E+00
Undefined                1           0         0.000000000E+00
--------------------------------------------------------------
```

Mass flow rate
Simulation: 125 kg/s
Theoretical: 125 kg/s

7.0 Conclusions

1. Modeling in SALOME is easy to handle, and it has got all the features when compared with other commercial modeling tools like CATIA, Pro/E, Solid Works etc.
2. Meshing process is very quick and mesh quality is well maintained, which can be successfully run for simulation. Any complex geometry can be meshed automatically using Netgen 1D-2D-3D algorithm, which distinguishes SALOME with other CFD tools.
3. SALOME/Saturne is a freeware tool, where different types of analysis can be carried out without much effort and quality of results are well maintained.
4. Since SALOME/Saturne is a freeware software's, it will give a large monetary benefit in terms of licensing cost.
5. A minimum hardware requirement to install this tool, further results in monetary benefits.
6. SALOME is compatible with large number of solvers including CAE_Linux, Solver Code Saturne, Openfoam, Elmer, Aster etc.

8.0 References

[1]. CAE Linux community portal, Contrib: Bond Matt/Laminar Pipe Flow
[2]. Salome-platform.org_user-section_salome-tutorials
[3]. Juha Kortelainen, Meshing tools for open source CFD-Apractical point of view-Research report-VTT-R-02440-09, 1 (24).
[4]. Gerry, Class notes_ME448-pipeflow tutorial, 2012
[5]. StarCCM+Version 8.04_ User guide, CD-Adapco.
[6]. Ansys-Fluent tutorial guide, Ansys Inc, November 2013.
[7]. Keith Martin and John M Simbala, Pointwise to Open foam tutorial-Laminar flow through a straight pipe, Penn state university, January 2011.
[8]. AVL FIRE tutorial, AVL-AST, Graz, Austria.
[9]. https:/www.cfd-online.com/Links/soft.html
[10]. Colominas, Paris and Fernandez et.al, A numerical simulation tool for multilayer grounding analysis integrated in an open source CAD-interface, International journal of Electrical power and Energy systems, September, 2012.
[11]. 11.Code_saturne-version2.0 tutorial, eDF, Fluid dynamics power generation and environment department, 2011
[12]. Peter Raback, Using Elmer with other pre and post processors, IT center for science, Elmer basic course, Finland, 2010.
[13]. 13.https:/www-code-aster.org/UPLOAD/DOC, Presentation/Plaquette_SALOME_V7.pdf