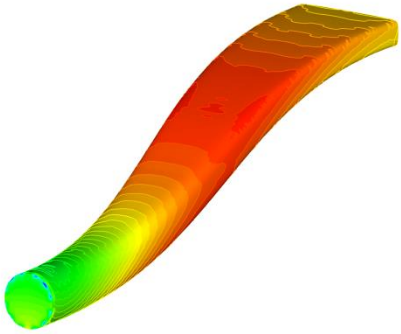
Coupling CAESES® and ANSYS® CFX®: S-Duct Example

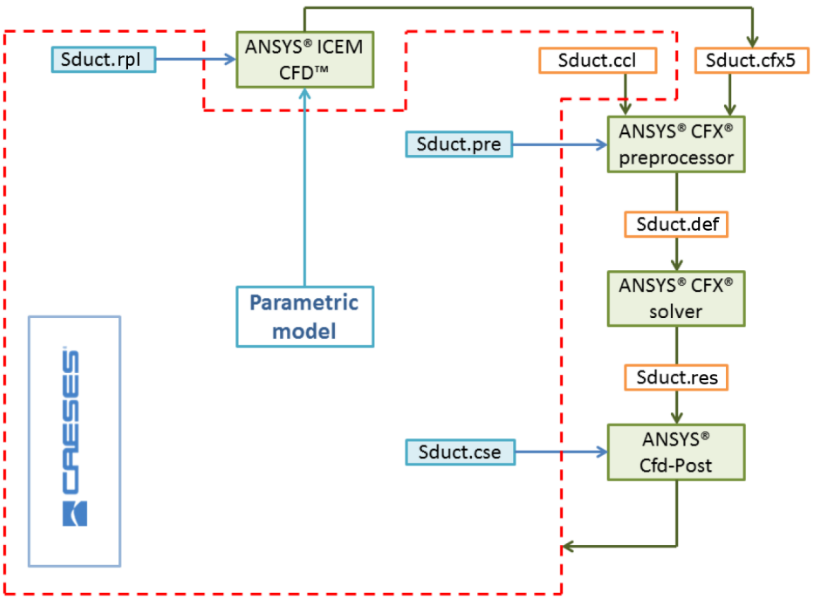
The purpose of this tutorial is to learn the integration of a 3rd party CFD software, in this case ANSYS® ICEM CFD™ and ANSYS® CFX®. You will use existing s-duct geometry, prepare it for the export and connect it with ANSYS®.

For the meshing setup we will use ANSYS® ICEM CFD™. While the simulation setup and the run will be performed with ANSYS® CFX® and for the post processing ANSYS® CFD-Post is utilized.



The s-duct geometry is created and linked to the software connector interface in CAESES®. Scripts for automating and controlling the whole process are created and then linked to the CAESES® software connector with the help of parameters. Finally, the simulation is executed, and the results are collected by CAESES® for post processing and optimization.

For any further questions you can use the CAESES® forum.



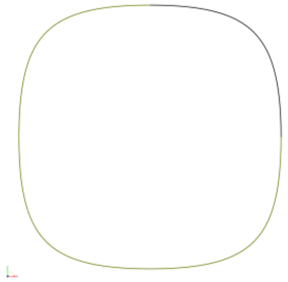
Open the S-Duct Model

The model from which we will start can be found in the documentation browser:

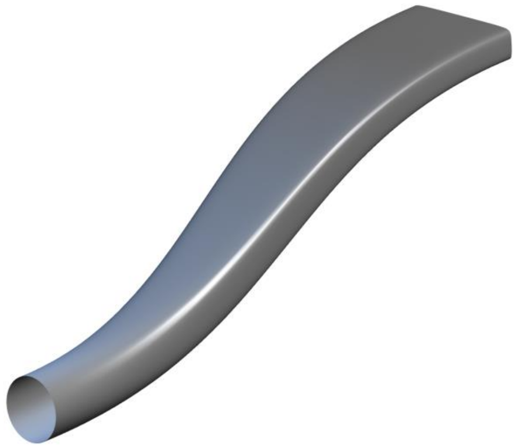
- Open the project *samples > design engines > s-duct geometry variation.*
- Save it as a new project.

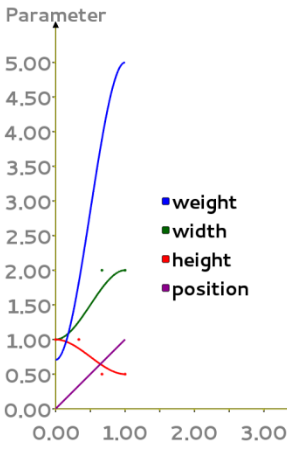
Here is a very brief overview of this model, before we continue with the ANSYS® CFX®. integration:

The first step of modeling the s-duct is to define the shape of the cross section. This is done inside the feature by using a NURBS curve with different transformations. The position of the cross section depends on the path. This feature definition will be the input for the curve engine. From the curve engine, a meta surface gets finally created, which sweeps the cross section along the path.



Different functions define the parameters of the cross section along the path, which are additional inputs for the curve engine. The functions are usually smooth B-Spline curves, where the x-position defines the curve path's parameter position (0 to 1) and the y-position the value of the parameter. These y-values can be controlled by design variables. You can move the points of the functions in y-direction, to see how the model changes.

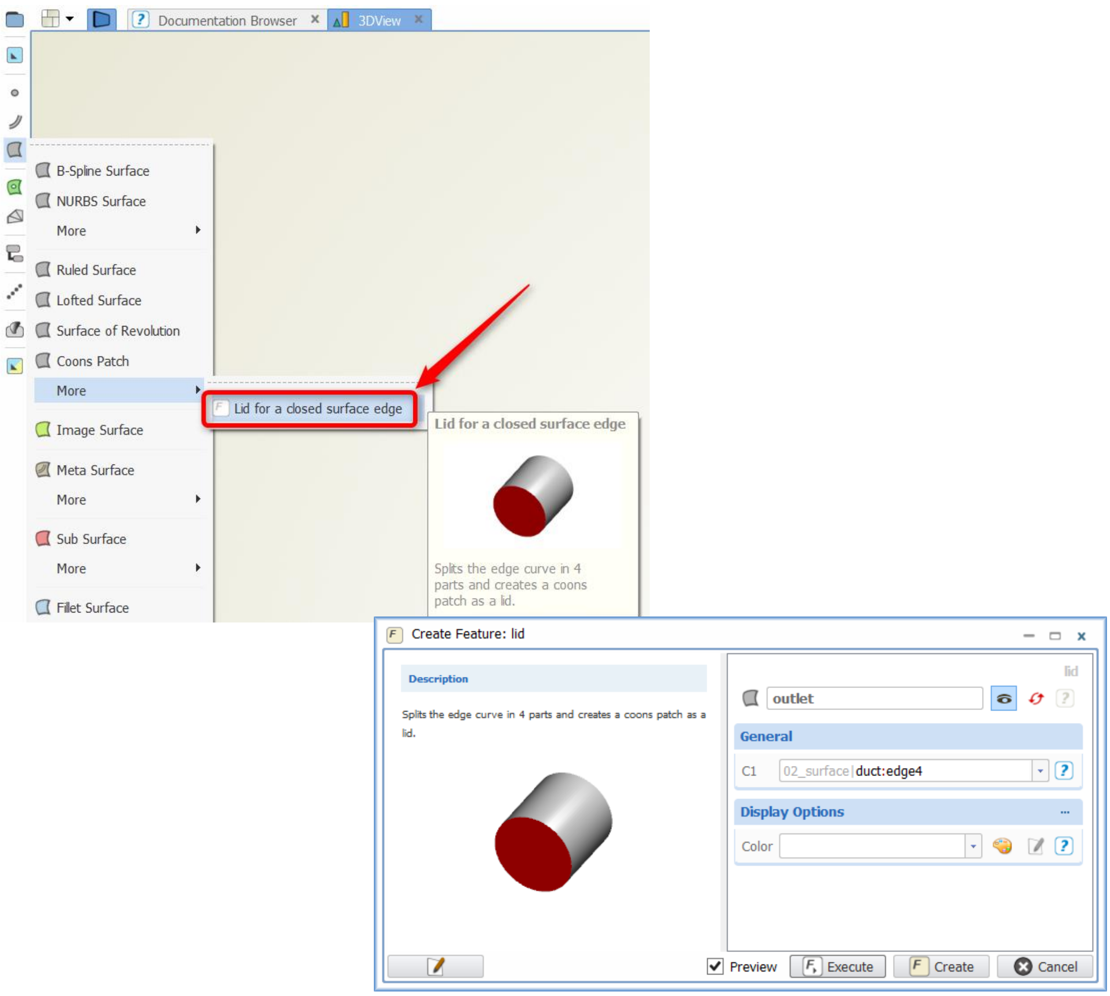




2 Prepare Geometry for Export: Lids

In order to export the geometry as a watertight CFD domain, we have to close the inlet and outlet and create a *brep*.

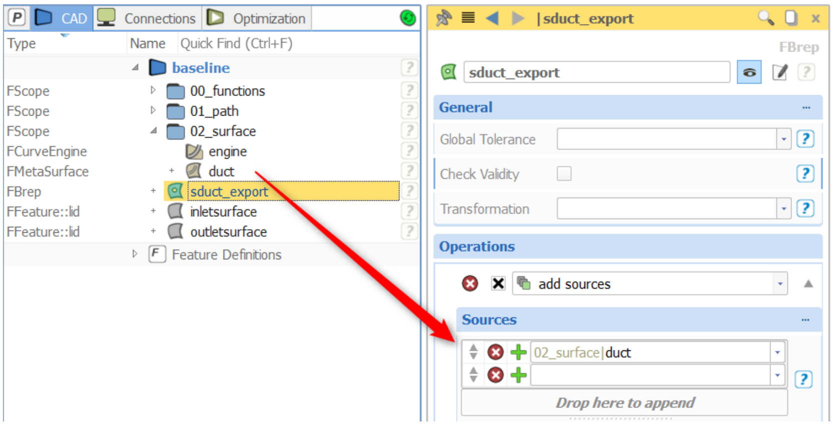
- Go to menu and select *CAD > surfaces > more (Coons Patch) > lid for a closed surface edge.*
- Rename the feature to “outletsurface”.
- Select the circular edge of the duct surface as the source for *C1.*
- Click “Create”. It is also possible to create the feature first, and then select the input curve.
- Repeat this step for the inlet surface naming it “inletsurface”.



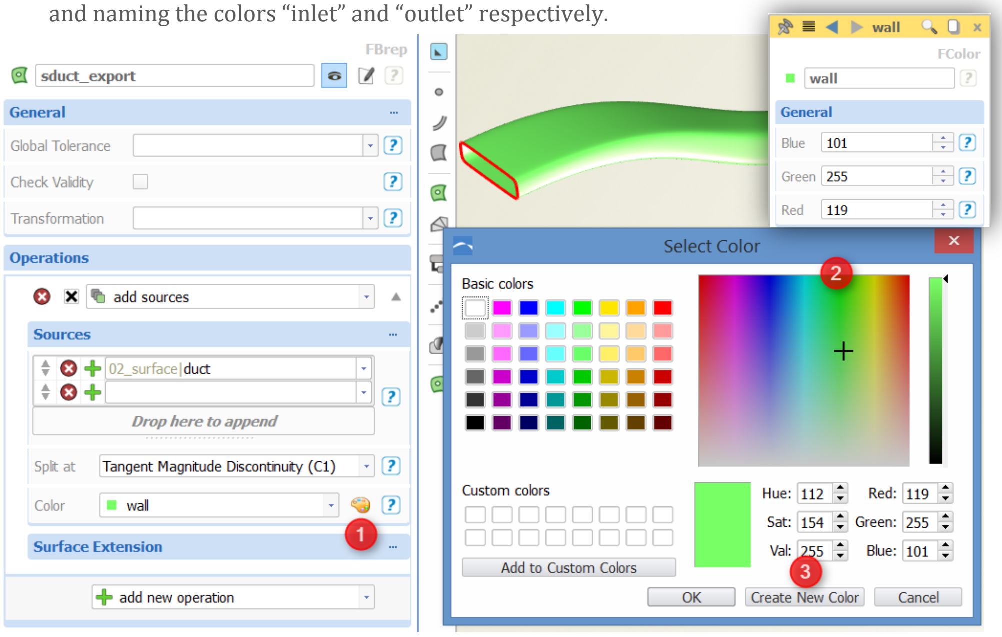
3 Prepare Geometry for Export: BRep

Now we can create a brep from the geometry components.

- Create a brep via *CAD > breps > brep* and call it "sduct_export".
- Under the *operations* menu, choose *add sources* and choose "duct".



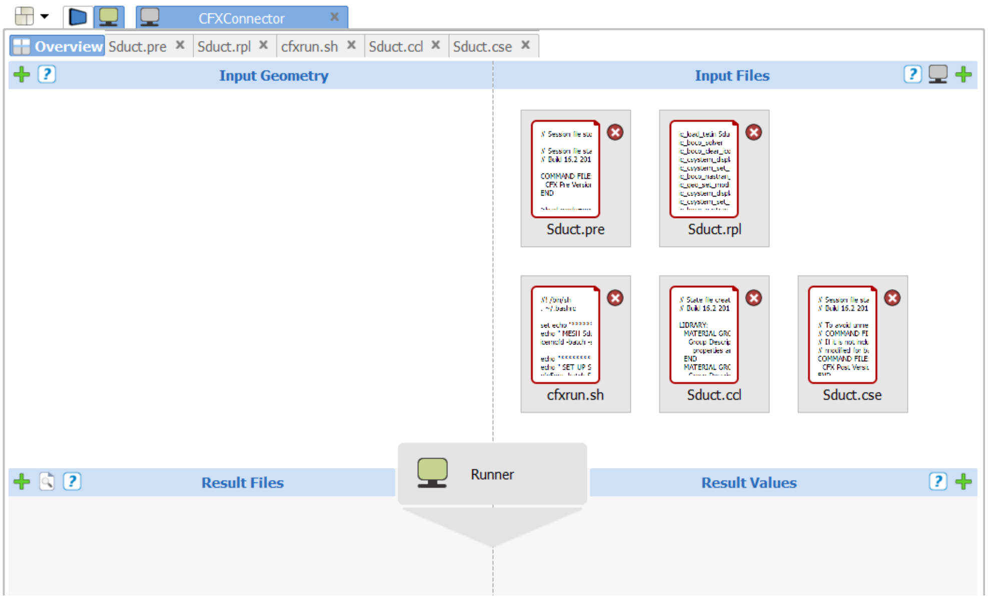
- Click on the "Create Custom Color" button and expand it.
- Choose the desired color and press "Create New Color" by renaming it, "wall".
- Enable the operation.
- Finally perform the previous four steps within the same brep both for "inlet" and "outlet" and naming the colors "inlet" and "outlet" respectively.



4 Import the Software Connector

Now we can import a prepared software connector which includes the ANSYS® ICEM CFD™, ANSYS® CFD-Post and ANSYS® CFX® files. It was configured before (for the purpose of this tutorial). The software connector is the widget where external tools can be plugged-in.

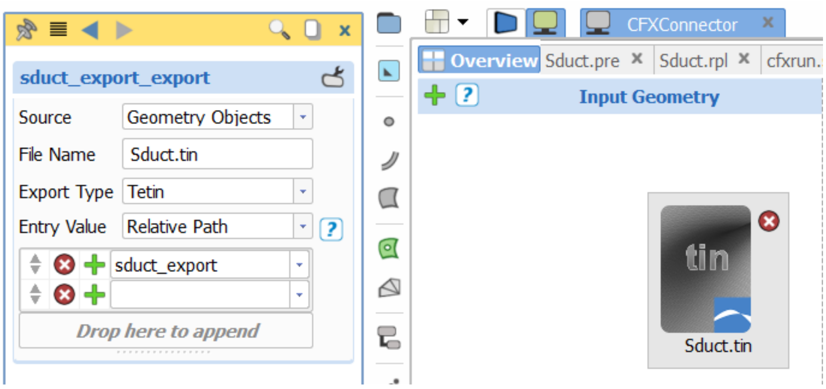
- Go to the *connections* tab in the pull down menu and choose "Software Connector From File".
- Select the file "06_ANSYS_Integration_softwareconnector.xffl" that you will find in the *installation directory > tutorials > 08_integrations.*



5 Software Connector: Input Geometry

Now we can begin to set up the input geometry in the software connector.

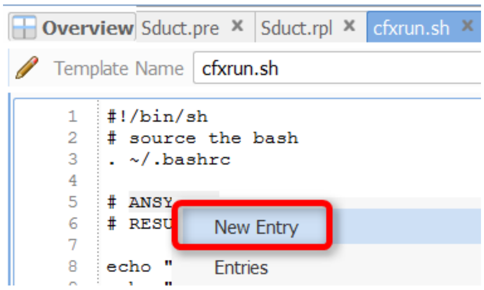
- Drag and drop the brep “sduct_export” into the *Input Geometry* window of the connector.
- Click on this file, and specify the *file name* to “Sduct.tin”.
- Set the *export type* to “Tetin”.



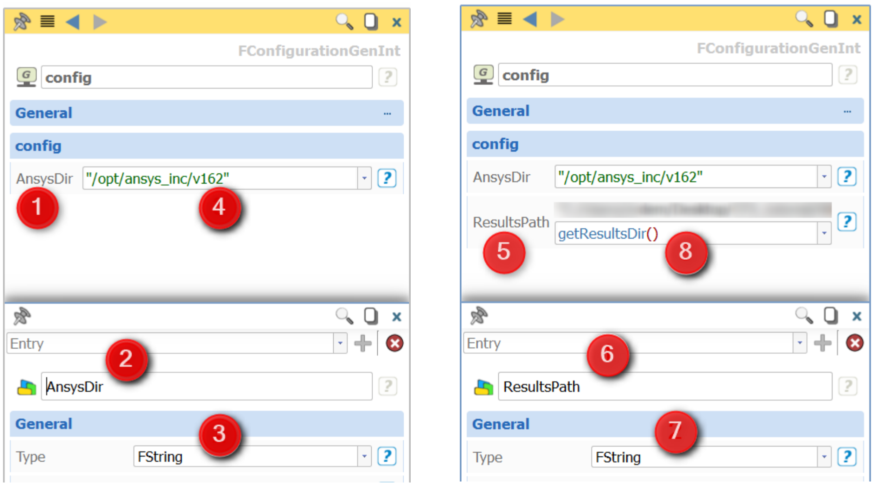
✓ The export file format Tetin (*.tin) for ANSYS ICEM CFD. With this export, the unique CAESES® IDs of edges and faces are directly transferred to ICEM CFD, and the user can rely on them when setting up a meshing process.

6 Software Connector: Set up Coupling

In this step, we will configure the software connector to trigger automatic simulation execution using these files.



- Open cfxrun.sh in the software connector.
- Highlight the "ANSYSDIR" in line 5.
- Create a new entry by right-clicking on the highlighted "ANSYSDIR" and select "New Entry".
- Click the name of the entry to edit its properties.
- Change the type to *fstring*, and rename it to "AnsysDir".
- Change the text in the field to your local ANSYS directory.
- Highlight "RESULTPATH" in line 6 and create a new entry.
- Rename it to "Resultpath" and change the type to *fstring*.
- Type getResultsDir() in the entry field, which returns the result directory for the current design.

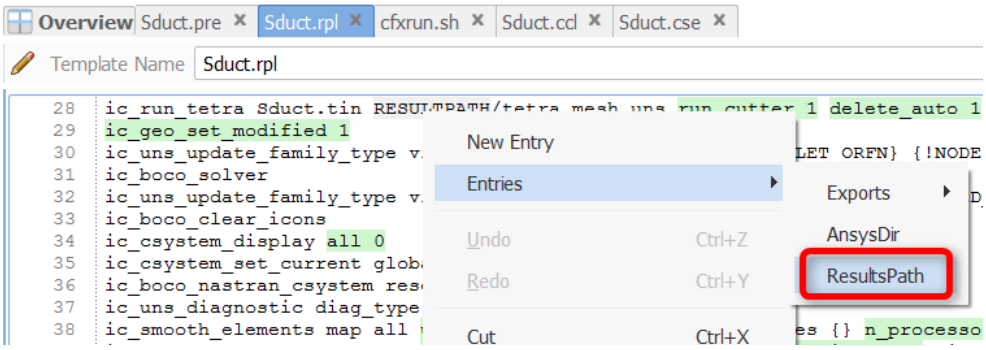


7

Parameterize the Input File Sduct.rpl

In this step we will edit the Sduct.rpl meshing input file by creating controls for mesh parameters.

- Find the working directory (“RESULTPATH“) and replace all with the “ResultsPath” entry.
- Find the ANSYS directory (“ANSYSDIR“) and replace all with the “AnsysDir” entry.



- Search “Sduct.rpl” for the part size parameters, “ic_geo_set_family_params ” (line 47) and create new entries for the boundary layer parameters on the surface “WALL” by moving the cursor over the appropriate numerical values and clicking the dialogue that appears. Do this for element height, *ehght* and number of layers, *nlay* (line 47 and 55).
- Finally check the file to make sure that paths leading to files exported by CAESES® are replaced by the ResultsPath entry, and that the filenames themselves are replaced by entries for any exported files.

8

Parameterize the Input File Sduct.pre

In this step we will edit the ANSYS® CFX® input file, Sduct.pre by creating controls for imported and exported files.

- Open Sduct.pre
- Find the working directory (“RESULTPATH“) and replace all with the “ResultsPath” entry.

Overview | cfxrun.sh | Sduct.rpl | Sduct.pre | Sduct.ccl | values.txt | Sduct_001.out | Sduct.cse

Template Name: Sduct.pre

```
# Session file stopped:  2016/03/09 13:58:57

# Session file started:  2016/03/09 13:59:06
# Build 16.2 2015.06.30-00.03-134402

COMMAND FILE:
  CFX Pre Version = 16.2
END

>load mode=new
> update

> gtmImport filename=<entry>ResultsPath</entry>\
Sduct.cfx5, type=Generic, units=m, genOpt= -n, nameStrategy= Assembly
> update

>importccl filename=<entry>Exports|Sduct.ccl</entry>, mode = replace, autoLoadLibrary = none
> update


>writeCaseFile filename=<entry>ResultsPath</entry>\
Sduct.def,operation=write def file
> update

> update

# Session file stopped:  2016/03/09 14:00:13

```

12 Parameterize the Input File Sduct.ccl

In this step we will edit the ANSYS® CFX® case file, Sduct.ccl by creating controls for boundary conditions.

- Open Sduct.ccl and search for the INLET and OUTLET boundaries.
- Create entries for inlet normal velocity and outlet relative pressure conditions. Do this for other simulation parameters, which you might think would be useful, such as number of iterations, convergence criteria, and timescale factor.

Overview | cfxrun.sh | Sduct.pre | Sduct.rpl | Sduct.ccl | Sduct.cse

Template Name: Sduct.ccl

```
        MASS AND MOMENTUM:
          Normal Speed = <entry>Speed</entry> [m s^-1]
          Option = Normal Speed
        END
        TURBULENCE:
          Option = Medium Intensity and Eddy Viscosity Ratio
        END
      END
    END
    BOUNDARY: outlet
      Boundary Type = OUTLET
      Location = OUTLET
      BOUNDARY CONDITIONS:
        FLOW REGIME:
          Option = Subsonic
        END
        MASS AND MOMENTUM:
          Option = Average Static Pressure
          Pressure Profile Blend = 0.05
          Relative Pressure = <entry>Pressure</entry> [Pa]
        END
```

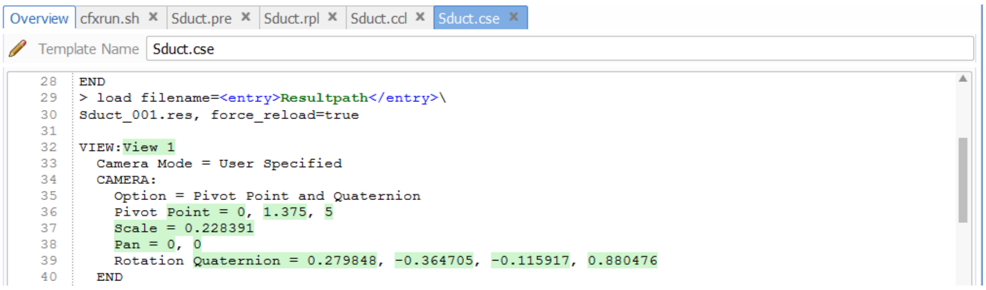
```
    CONVERGENCE CONTROL:
      Length Scale Option = Conservative
      Maximum Number of Iterations = <entry>Iterations</entry>
      Minimum Number of Iterations = 1
      Timescale Control = Auto Timescale
      Timescale Factor = 1.0
    END
    CONVERGENCE CRITERIA:
      Residual Target = <entry>Target</entry>
      Residual Type = RMS
```

✓ Once the simulation is started, it can be monitored manually using the CFX solve manager, alternatively, CFX can be configured to automatically monitor each run, or export the convergence history to CSV files which can be read by CAESES as results tables for each run.

9 Parameterize the Input File Sduct.cse

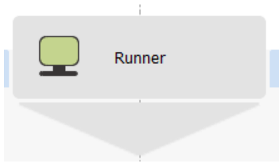
In this step we will edit the ANSYS® CFD-Post post process input file, Sduct.cse by creating controls.

- Open Sduct.cse
- Find the working directory (“RESULTPATH“) and replace all with the “ResultsPath” entry.

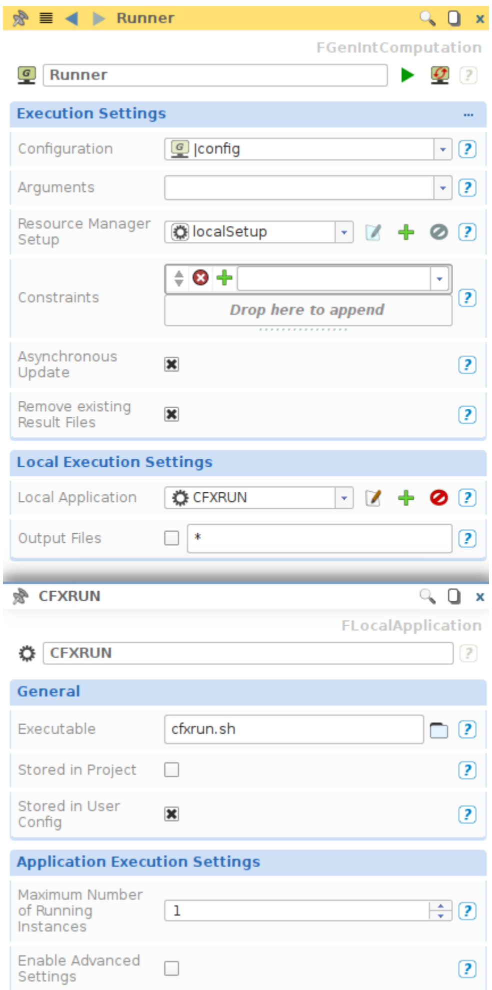


10 Setting up the Computation

For postprocessing, we need to have results files and values, which will be inserted in the *Results Files* and *Results Values* window of the software connector. In order to get these, we have to trigger a first run – either with CAESES®, or externally. Since we have already changed parts of the input files in the previous steps, we want to use CAESES® to run ANSYS® CFX®. Therefore we have to set up the *computation* "Runner":



- Click on "Runner" in the center of the *software connector*.
- In the object editor of the selected object "Runner", activate "Remove existing Result Files". This makes sure that with each run the previous simulation files get deleted automatically, and is similar to the "Allclean" command.
- Create a new "Local Application" by clicking on the plus-button next to the attribute, and call it "CFXRUN".
- As executable write "cfxrun.sh". This will trigger the "cfxrun.sh" script.
- Run the computation by clicking on the green run button (▶).
- While the computation is running, you can check the output in the *Task Monitor*.



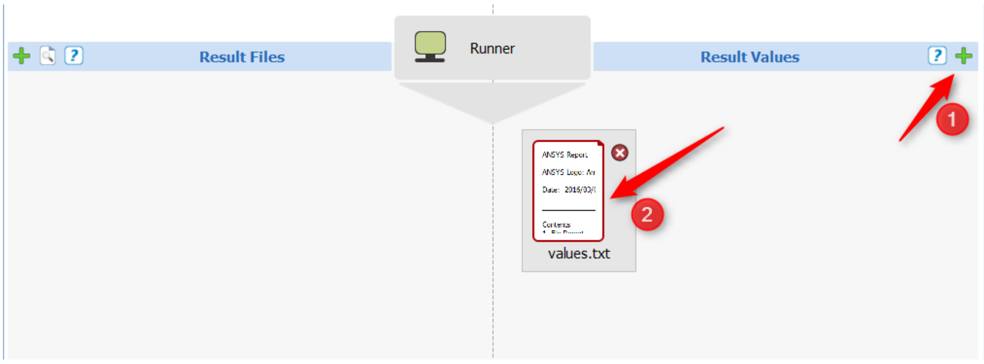
When the computation is done, check the results on your hard disc, to see how the results are handled by CAESES®: A new folder was created, with the name of the current project file (*.fdb). In this folder all results can be found.

11

Result Values Part 1

In order to assess the simulation, we need to have result files which provide for example the pressure drop. These values can be extracted from any text files.

- In your file explorer, go to the baseline directory which was created during the last run and check for the “values.txt” file.
- Add the “values.txt” file to the *Results Values* window of the software connector by using either drag & drop, or by using the green plus button at the window.
- Double-click on “values.txt” in the software connector to see the content.

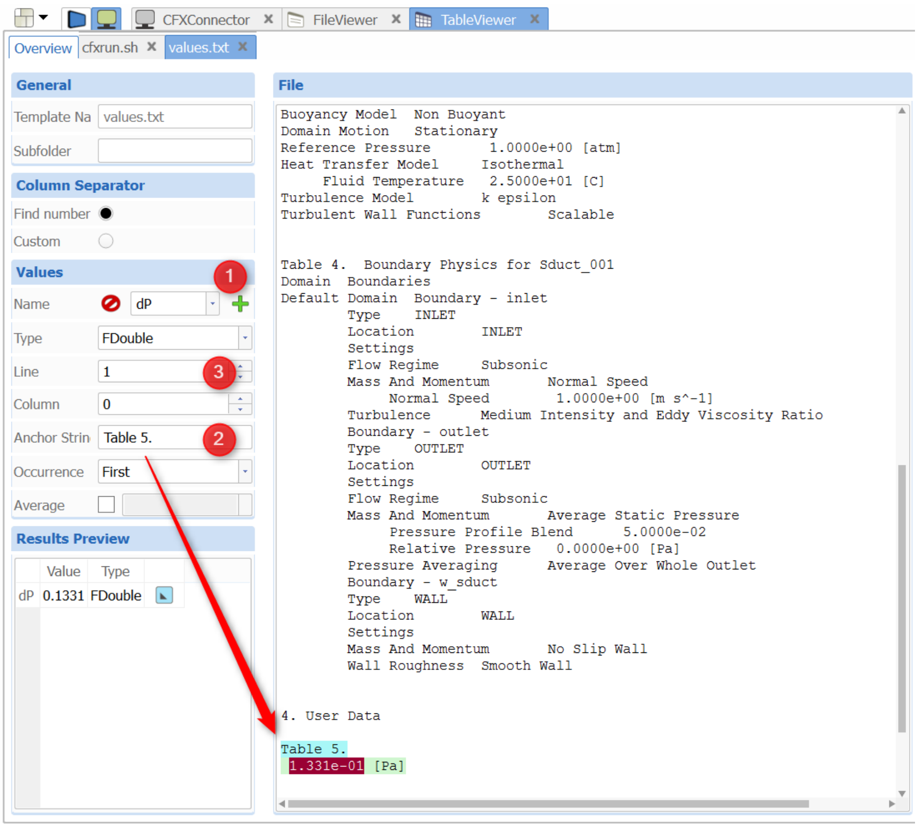


12

Result Values Part 2

After importing the template result file. We are now able to extract result values such as the pressure drop.

- Add a value by clicking on the plus button and name it “dP”.
- In *Anchor String* part add “Table 5.” where CAESES® will be searching from this point on.
- Set *line* to “1” (in this case, this means: always the last row of the file).
- Set column to “0”.
- Now create a parameter for the value by clicking on the blue parameter symbol in the results preview table. This parameter will be the objective of the simulation at a later stage.
- Select the evaluation parameters, and create a scope. Set the name of this scope to “CFX_evaluation” (note, names can be user-defined i.e. arbitrary).

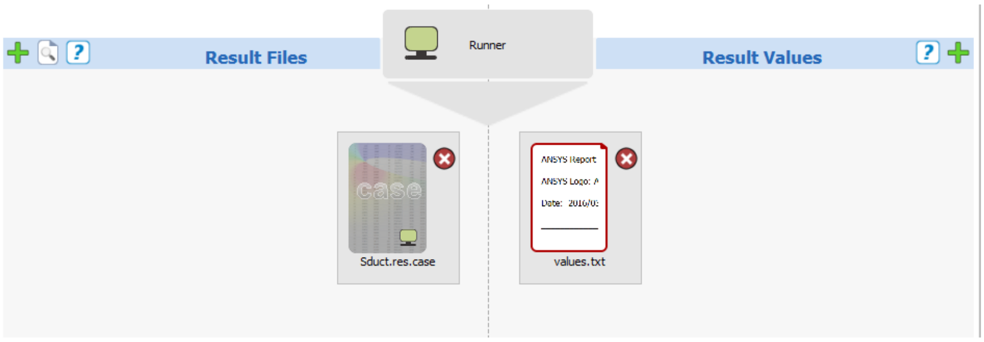


✓ Remember that this parameter is the result value of the last iteration or time step. If you have a strong oscillating result, you might average the values over a specific time.

13 Result Files

In the *Result Files* window of the connector, output files from the CFD calculation are referenced. Typical output files are pictures, tables, text files and CFD solutions.

▶ Add the "Sduct.res.case" Ensight file to this window, either by using drag & drop, or by using the plus button in the corner.

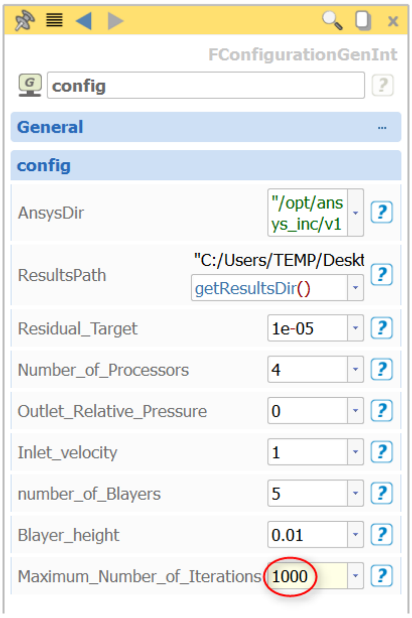


14

Running the Case

Now the software connection is completed, and a first test case can be started.

- Set the parameter *Maximum_Number_of_Iterations* to “1000”.
- You can also decrease the “Residual_Target” factor, which will increase the computational time.
- Go back to the software connector and run the simulation (again by using the ▶ button).



Appendix

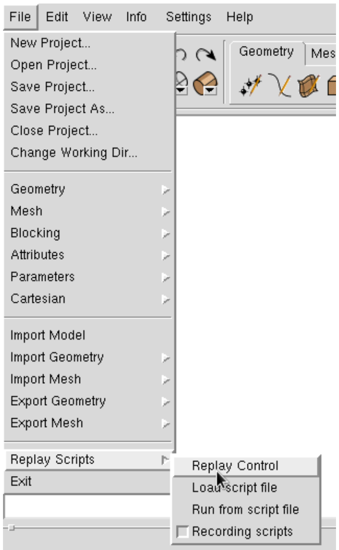
This appendix will guide you through the process of creating your own ANSYS script files.

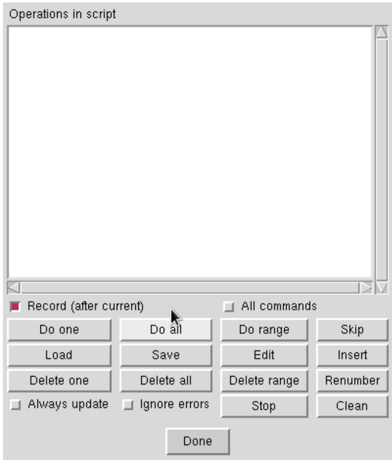
A1

Meshing with ANSYS® ICEM CFD® Part 1

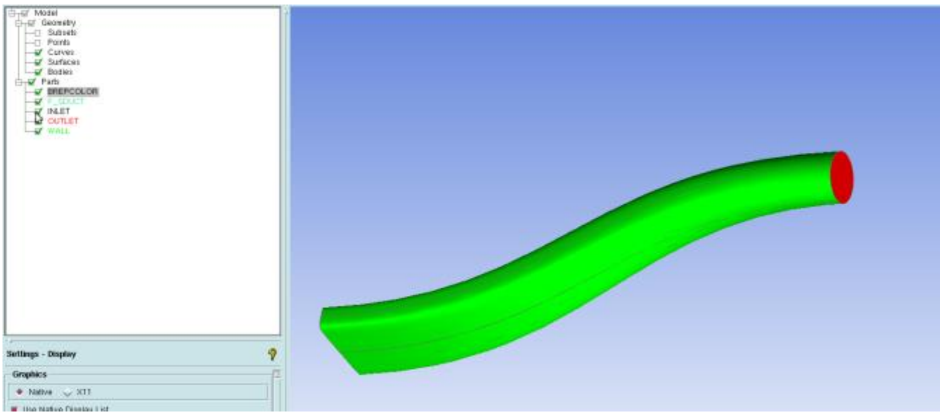
In this section, a script file is created using ANSYS® ICEM CFD® to automate basic unstructured meshing of the flow domain.

- Open ANSYS® ICEM CFD®.
- Begin recording scripts by selecting “File > Replay Scripts > Replay Control”. Ensure that recording scripts is checked





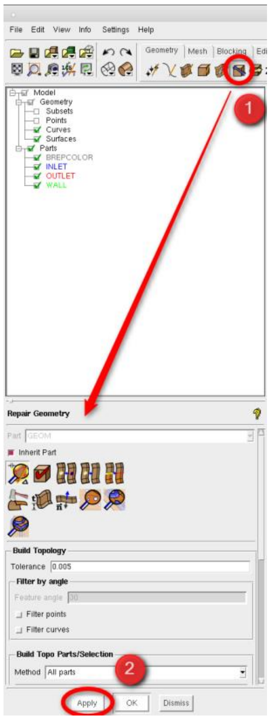
- Change the working directory of ICEM to the working directory by selecting “File > Change Working Dir…”
- Import the geometry file by following the path “File > Geometry > Open Geometry” and selecting Sduct.tin as the import file.



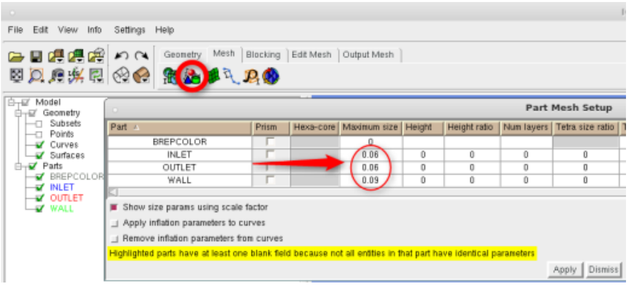
A2

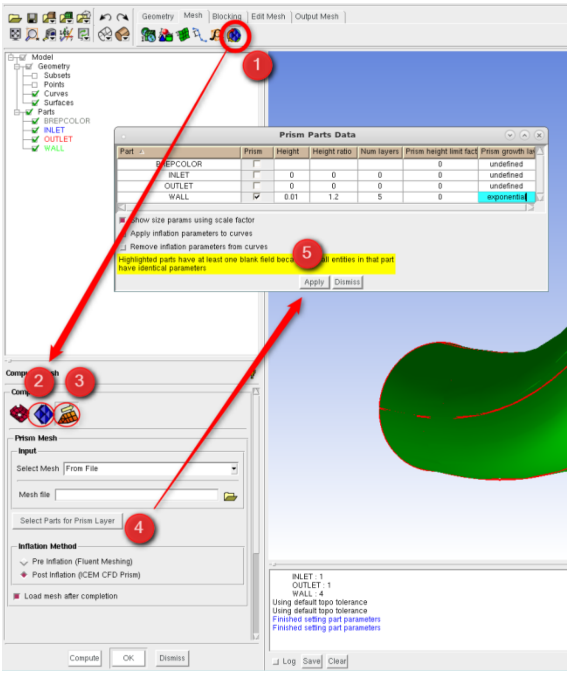
Meshing with ANSYS® ICEM CFD® Part 2

In this step we are going to continue to create the meshing macro.



- Under Geometry tab, pick Repair Geometry and click OK in the opening menu.
- Under the mesh tab, bring up the part mesh-setup dialogue, and input size parameters for all surfaces imported using the values indicated. Leave all other parameters as zero.





- Create an unstructured volume mesh by bringing up the Compute Mesh dialogue box under the Volume Mesh tab.
- Create a boundary layer under the Prism Mesh tab.

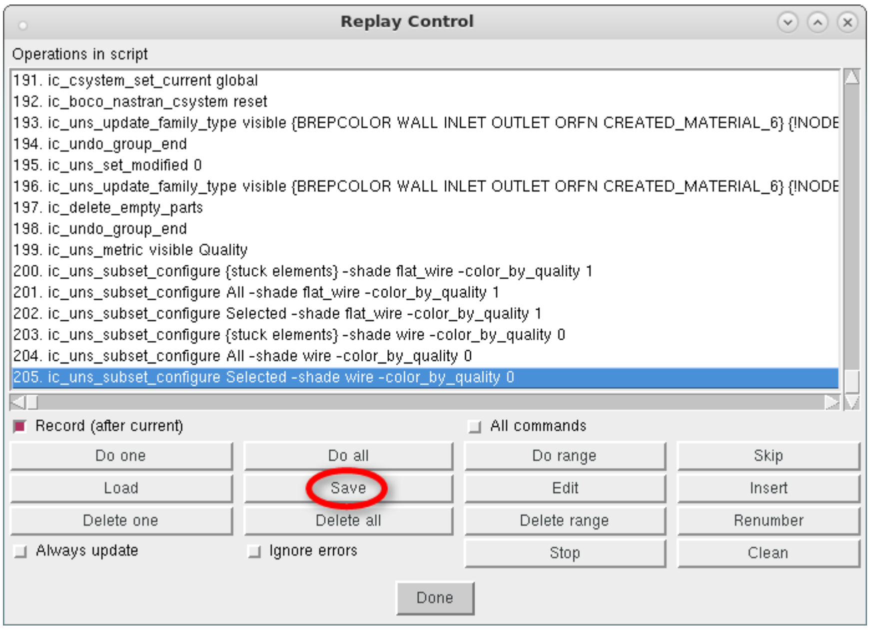
When the mesh is finished, inspect the prism mesh and split the elements if desired.

A3

Meshing with ANSYS® ICEM CFD® Part 3

In this step we are going to continue to create the meshing macro.

- Click through the “Settings > Solver menu” to bring up the solver setup dialogue and specify ANSYS® CFX® as the solver type.
- Click on the output tab and select write input to write the mesh files, save the project when prompted to the working directory.
- Ensure that the .boco file is also saved in the working directory, and then export the files.
- In the replay control window, select “Done” and save the replay in the working directory as “Sduct.rpl”.
- Close ANSYS® ICEM CFD®.



✓ Group parts by giving them identical spacing, which later you can easily find by searching for the spacing value and link to a common entry in the software connector

A4

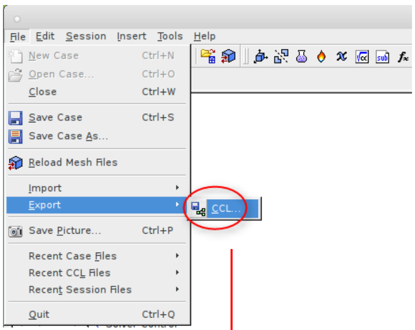
Set up the CFX Case file Part 1

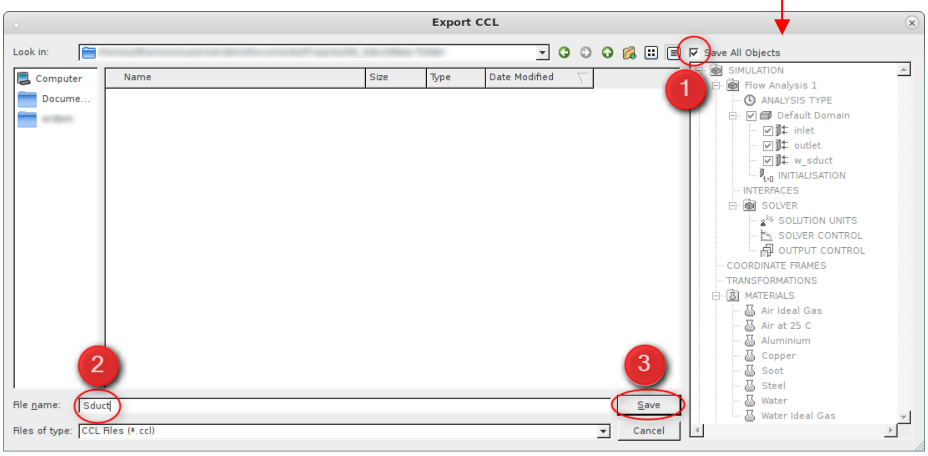
In this step we will create a template simulation setup and a script file using CFXpre to automatically generate a simulation setup and export the solver definition file.

- Open CFX and create a new project, importing the mesh generated in the previous section.
- Set up the simulation case as desired or as is necessary. Some basic sample boundary conditions are:

Boundary Name	Boundary Type	Details
inlet	Inlet	Normal velocity: 1.0 m/s
outlet	Outlet	Zero average gauge pressure
wall	Wall	No slip - Rotating

- Ensure to set up an execution control object, this will be important for defining the solver working directory.
- Export the case setup from the menu by browsing through File > Export > CCL, naming the file "Sduct.ccl", and ensuring to select "save all objects" to export the entire setup.
- Close the project



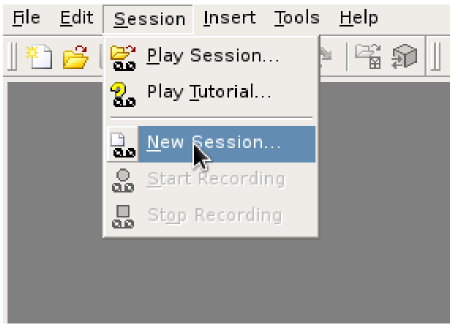


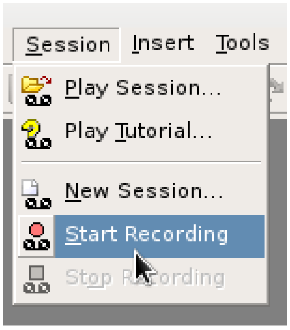
A5

Set up the CFX Case file Part 2

Follow up of the step before.

- Create a new session file, naming it Sduct.pre and save it to the working directory, then select “Start Recording”.





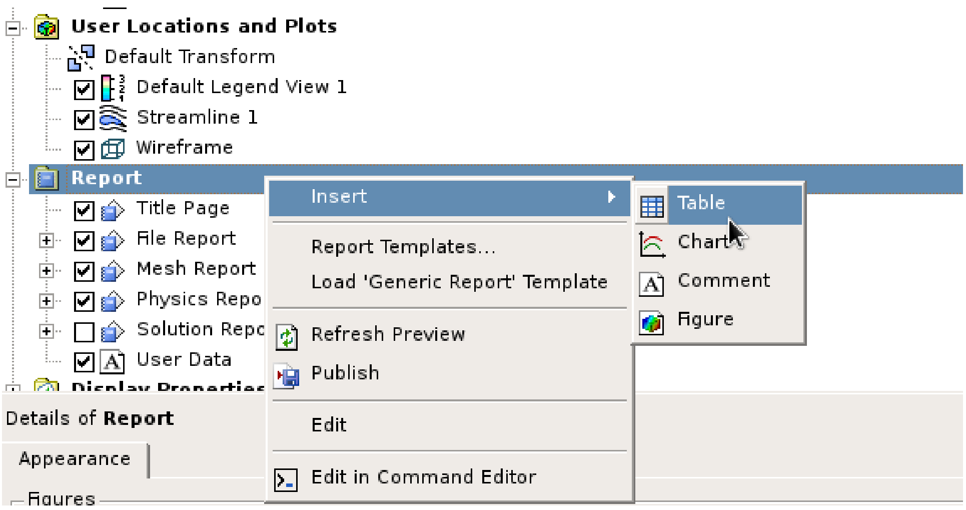
- Import the mesh file Sduct.cfx5.
- Import the Sduct.ccl file created.
- Write a solver input file to the working directory.
- Stop recording and close CFX, saving the project if desired.

Set up Basic Post Processing and Results Part 1

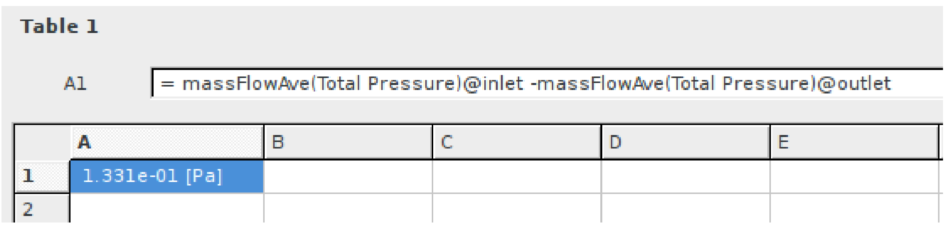
A6

Finally, a script file is created to automate post processing in CFX Post, and integrated into CAESES to automate the post processor execution. Results are imported into CAESES for post processing and optimization.

- Open CFXpost and start a new session, saving it to the working directory, and naming the script file "Sduct.cse".
- Start the recording.
- Create a table in CFX post to compute the pressure difference between the inlet and outlet boundaries, and then export it as a txt file in the working directory.
- Create a new table in the report section of the CFX Post object tree by right-clicking on report heading, and selecting "Insert > Table".



- In the first cell, enter the following equation: "=massFlowAve(Total Pressure)@inlet -massFlowAve(Total Pressure)@outlet".



A7

Set up Basic Post Processing and Results Part 2

Follow up of the post processing and results set up.

- Uncheck all items in the report subheading except the table created in the previous step, then right click on report and select publish.
- Select "Text" as the Export format, and then export the file to the working directory.
- Stop recording.