C10

SIMULATION OF COMPRESSIBLE INTERNAL FLOW SYSTEMS WITH ECOSIMPRO

José Javier Álvarez García – Industría de Turbopropulsores, S.A.
Parque Empresarial San Fernando de Henares, 28830 Madrid - Spain
josejavier.alvarez@itp.es

Abstract

A library of typical internal flow components has been developed based on the EcosimPro simulation tool. Each of the components is characterised by the basic fluid mechanics equations (flow conservation, momentum and energy) that link component inlet and outlet variables. Empirical data and correlations from reliable sources are mostly used to account for momentum and energy losses in components. The object oriented approach EcosimPro provides makes it suitable for the creation of increasingly complex internal flow networks which may eventually accurately represent actual air systems of any kind. Component equations as devised can only account for steady flow processes. The possibility of using time as an additional parameter in EcosimPro allows for building quasi-steady flow processes which may support parametric studies of any kind.

1 INTRODUCTION

In the world surrounding aircraft engines and turbomachinery design and development a proper understanding of internal compressible air flow is a need. It is also a must to be capable of simulating correctly compressible air systems. The aeroengine main flow path itself is an internal compressible flow system; the engine secondary air system (SAS), consisting of bleeds, seals and ducts to drive cooling and sealing air into bearing chambers and back into the turbomachinery main flow path is also a good example.

Furthermore, extensive turbomachinery rig testing is required during component (compressor or turbine) development. Rig facilities for these purposes indeed require dealing with compressible flows. A correct simulation of the flow is vital for developing facility control laws and for pre-determination of test conditions.

This paper deals with the simulation, using EcosimPro, of internal air flow components extensively used in turbomachinery rigs as well as other generic components which may be used to represent localised losses in aeroengine secondary air flow systems.

Before going into further detail on specific component simulation a description of the general governing equations of compressible air flow will be firstly made, including the simplifying assumptions adopted. One-dimensional steady flow will be assumed throughout and the air will be dealt with as an ideal, calorifically semi-perfect gas, allowing for variations with temperature of the air specific heat only.

Secondly, a description of the structure of the component flow port is made also describing their interrelations with the main bodies of the components.

A description of the most relevant features of the overall model is then made with a specific reference to: a) different sets of boundary conditions that may successfully be assigned to the flow networks created, based on library components; b) zero-flow solutions; c) supersonic flow solutions discard and flow choking; d) selection of suitable tearing variables.

An overview of the most representative components is eventually made as an illustration of the basic structure of the various components.

2 GOVERNING EQUATIONS AND ASSUMPTIONS

It is not the aim of this paper to provide a detailed description of the compressible fluid flow theory. However, the structure of the equations used in the model has an strong impact on ECOSIM capability to solve the system.

One-dimensional flow theory is used throughout, such that a unique set of mean value variables is generated at every component port. Fully hydrodynamically and thermally developed flow is assumed at all components. Two or three dimensional flow effects are regarded to be accounted for via the various data and correlations used in the components.

Most of the data and correlations that may be found in the bibliography consider ideal and calorifically perfect gas theory (considering specific heat ratio variations just in a few cases). For the sake of congruence air has been considered herein as an ideal and calorifically semi-perfect gas.

For a given section of an internal one-dimensional flow, the following equations apply:

$$c_P = c_P(T)$$

$$\gamma(T) = \frac{C_P}{C_P - R}$$

$$\left(\frac{G}{A}\right) = \left(\frac{G}{A}\right)^* \cdot F(M)$$

$$\frac{T_t}{T} = (1 + \frac{\gamma - 1}{2} \cdot M^2) \qquad [1]$$

$$\frac{P_t}{P} = (\frac{T_t}{T})^{\frac{\gamma}{\gamma - 1}}$$

$$V = M \cdot \sqrt{\gamma \cdot R \cdot T}$$

$$\rho = \frac{P}{R \cdot T}$$

$$H_t = G \cdot C_P \cdot T_t$$

where,

G = mass flow (kg/s)
A = sectional area (m^2)
ρ = density (kg/m^3)
R = air constant =287.01 J/kg/K
P = static pressure (Pa)
P_t = total pressure (Pa)
T = static temperature (K)
T_t = total temperature (K)
H_t= total enthalpy (W)
M = Mach number
γ = air specific heat ratio
$\left(\frac{G}{A}\right)$= flow per unit area ($kg/s/m^2$)
$\left(\frac{G}{A}\right)^*$ = critical flow per unit area ($kg/s/m^2$)

The critical flow per unit area represents the flow per unit area at choked flow conditions, i.e. when flow Mach is unity, and is a function of stagnation pressure and temperature. It is also a function of static temperature through the value of γ. For the sake of simplicity in the calculations it has been assumed in the model that the specific heat properties of the air are functions of the stagnation temperature rather than static temperature. The expression for critical flow per unit area is shown in equation [2].

$$\left(\frac{G}{A}\right)^* = P_t \cdot \sqrt{\frac{\gamma}{R \cdot T_t}} \cdot \left(\frac{\gamma + 1}{2}\right)^{\frac{\gamma + 1}{2 \cdot (1 - \gamma)}} \qquad [2]$$

This approach yields more accurate values than an ideal gas approach (constant specific heat properties) provided that flow temperatures are moderate or high and flow velocities are relatively low which is the usual case in turbomachinery test rigs as well as engine SAS.

F(M) is a function of the Mach number and of static temperature through the value of γ (now simplified to stagnation temperature), and is given by the following expression, in equation [3]:

$$F(M) = M \cdot \left[\frac{2}{\gamma + 1} \cdot \left(1 + \frac{\gamma - 1}{2} \cdot M^2\right)\right]^{\frac{\gamma + 1}{2 \cdot (1 - \gamma)}} \qquad [3]$$

F(M) represents the existing ratio between the actual flow and the critical flow for the stagnation pressure and temperature values considered.

Analysis of equations [1] – [3] shows that three variables need to be fixed to obtain the flow state at a specific section for a fixed geometry.

3. COMPONENT FLOW PORT AND COMPONENT BASIC STRUCTURE

3.1. COMPONENT FLOW PORT

The flow port used for the components is described below:

```
PORT airflow
   EXPL REAL Cp RANGE 0, Inf -- Specific heat at constant pressure [J/kg/K]
   EXPL REAL gamma         RANGE 0, Inf -- Specific heat ratio
   EXPL REAL Gac RANGE 0, Inf -- Critical mass flow per unit area [kg/s/m2]
   SUM REAL G RANGE 0, Inf -- Mass flow [kg/s]
   SUM IN REAL Ht RANGE 0, Inf -- Total enthalpy [W]
   EQUAL OUT REAL Tt RANGE 0, Inf -- Total temperature [K]
   EQUAL REAL Pt RANGE 0, Inf -- Total pressure [Pa]
CONTINUOUS
   Cp = Cp_T(Tt)
   gamma = Cp / (Cp - Rair)
   Gac = Pt * sqrt(gamma / Rair / Tt) * ((gamma + 1) / 2)** \
        ((gamma + 1) / 2 / (1 - gamma))
   Ht = G * Cp * Tt
END PORT
```

The port considers three main variables, G, P_t and T_t, and some others which may be directly derived from them, i.e. C_P, γ, Gac (critical flow per unit area) and H_t. These derivations are obtained via the Continuous part in the component.

G is only a transfer variable with no functions applied to it because neither M nor A appear in the port. The flow area is needed to obtain the actual flow when the stagnation pressure and temperature are known (M can be determined once the others are fixed). However flow area may only be accessible from component data. This leads to two different approaches to tackle the problem: a) the solution adopted consisting in matching G for datum geometry A by varying M inside each component; b) an alternative approach consisting in including M in the port solving in its continuous part for a value of flow per unit area. Actual flow G would then be internal information in the component once the equation is solved by considering the geometry A.

Although the latter alternative approach requires less code and it is thus probably faster to solve, the former approach has been followed as it offers the advantage that geometry information is not strictly required at each port. This yields more a flexible structure as it is capable to deal with contraction/expansion processes from/to plenum chambers where geometry information is not relevant. On the other hand, it does not introduce flow discontinuities when solving for incorrectly input geometry data.

3.2 COMPONENT BASIC STRUCTURE

A typical example of basic calculation inside a component is shown below. It consists first of the set of equations which link the inlet and outlet ports of the component: conservation of flow, momentum and energy equations. Flow is calculated at the inlet and outlet with the values of areas, critical flows per unit area and F(M) functions, iterating over the Mach numbers, and both flows are then made equal. The momentum equation is represented in this case as a stagnation pressure ratio variable which will generically be a function of various derived and direct port and component variables. The energy equation will generally follow the same approach followed for the momentum equation.

Other miscellaneous flow information such as static pressure, static temperature, static density, velocity, etc) is obtained via a dedicated function which requires P_t, T_t and M as inputs. The equations generated for the stagnation pressure and temperature ratios for each individual component will obviously conform the actual difference among components.

```
-- Links between bend inlet and outlet
out.G = in.G
in.G = A1 * in.Gac * fM(in.gamma, in_M)
out.G = A2 * out.Gac * fM(out.gamma, out_M)
out.Pt = in.Pt * PRt
out.Tt = in.Tt * TRt

-- Calculation of fluid data
f_data (in.gamma, in.Pt, in.Tt, in_M, in_P, in_T, in_rho, in_V,
in_dtf)
f_data (out.gamma, out.Pt, out.Tt, out_M, out_P, out_T, out_rho,
out_V, out_dtf)
```

```
FUNCTION NO_TYPE f_data (
  REAL gamma,         " Specific heat ratio [J/kg/K]"
  REAL Pt,            " Total pressure [Pa]"
  REAL Tt,            " Total temperature [K]"
  REAL M,             " Mach number"
  OUT REAL P,         " Static pressure [Pa]"
  OUT REAL T,         " Static temperature [K]"
  OUT REAL rho,       " Density [kg/m3]"
  OUT REAL V,         " Velocity [m/s]"
  OUT REAL dtf        " Dynamic temperature factor" )
BODY
  dtf = 1 + (gamma - 1) / 2 * spow2(M)
  T = Tt / dtf
  P = Pt / dtf**(gamma / (gamma - 1))
  rho = P / Rair / T
 V = M * ssqrt(gamma * Rair * T)
END FUNCTION
```

4. OVERALL MODEL FEATURES

4.1. BOUNDARY CONDITIONS

All of the components generated have been made one-directional, i.e. only forward flow is allowed. This has been necessarily imposed, at least at this stage of the development, as many components present asymmetric behaviour (e.g. a diffuser would behave as a nozzle with backward flow; a compressor cannot behave as a turbine) and completely different modelling would be required. Future development will try to unify different behaviours for similar geometry components wherever possible.

A flow network made of several components would thus require, by default, three boundary conditions at the inlet port of the very first component: G, P_t and T_t. This conforms to one of the typical calculations required by analysts and designers, i.e. for a flow through a system known, calculate the pressure loss incurred. The other typical calculation required is also possible for this structure, i.e. for a known pressure differential across a system, calculate the flow obtained. This would require swapping the boundary condition of the first component inlet G by last component outlet P_t. It should be noted the latter procedure brings about a huge system equation box as all components are coupled which is not the case for the former procedure.

Design options are also possible with EcosimPro in certain specific cases, although there is a fallback solution for those case where it was not possible, consisting in doing a parametric study with the selected data.

4.2. ZERO FLOW SOLUTIONS

Infinite resulting from zero flow solutions, e.g. proportional or discrete valves closed or equal differential pressure imposed on a component or network, where some of the variables have their values at null, e.g. Mach number, Reynolds number, velocity, require special attention when modelling.

The critical flow per unit area in equation [2] within the component port is forced to be solved explicitly, which causes the solution on flow to be obtained in the components by making the critical flow times the area and times the Mach number function *F(M)*. Flow is then always obtained explicitly by iteration on M until flow continuity is achieved. Zero flow solutions makes M null but no division by zero is incurred. Equations to obtain the momentum link between the inlet and outlet ports of the components are always introduced such that M never appears dividing, which is sufficient a requirement to achieve defined solutions as M is always force to be a tearing variable.

Other variables such as Reynolds number, Re, or Dean number, Dn, become zero when flow is null. This may introduce indetermination into some variables based on them, e.g. the laminar flow friction factor, obtained as

$$f = \frac{64}{\mathrm{Re}}$$

Different approaches could be used to override this potential problem, the one used being limiting variables to a certain threshold level, sufficiently low not to cause instabilities, e.g. always Re >1.

The solution of zero flow conditions introduced by proportional valves when closed is a different issue that needs special treatment. When valves close, the momentum equation becomes uncoupled from the flow conservation equation, i.e. a pressure differential across the valve may exist even if no flow exists. In fact, if a pressure differential is imposed on the valve, the valve chokes (see paragraph 4.3) when its opening is small enough. Valve choking uncouples those two equations too. Imposing a certain amount of flow as a boundary condition in the valve is obviously incompatible with closed valve conditions, as is in the case choking flow occurs in an in-series component line within a network.

When flow is not set as a boundary condition in the valve, the way around the problem would consist in solving for the Mach number at the inlet to the valve opening orifice, Ma, based on pressure differential (or pressure ratio) conditions through the valve, which would be related to a guess of the upstream valve free stream conditions. Whatever valve of Ma obtained would be overridden by the value of valve opening area, which would be set to zero if the valve is closed. This would yield a zero flow condition at the valve outlet port independent of Ma. Inlet valve free stream flow equation will iterate on inlet port M till the zero flow condition is satisfied. A regulating valve is included as an example in next chapter.

4.3 SUPERSONIC FLOW SOLUTIONS DISCARD AND FLOW CHOKING

The function *F(M)*, which relates actual mass flow to critical mass flow (see equation [3]), is represented in Figure 1.

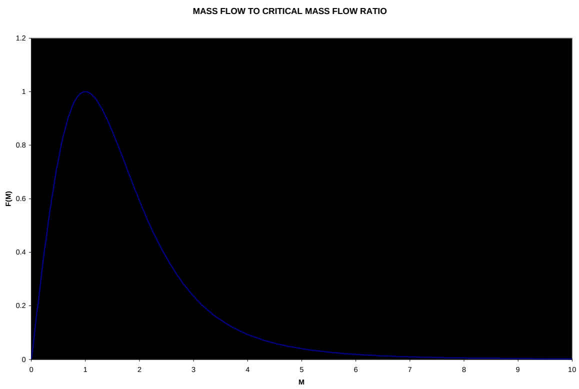

Figure 1: Mass flow to critical mass flow ratio

As it can be seen, it presents a maximum at Mach number unity of value unity, i.e. critical mass flow is reached at sonic conditions (M = 1). As the Mach number is used at all ports as a tearing variable, it may well be the case that a solution to the equations is found by the solver in the right hand part of the curve, i.e. a supersonic solution, unrealistic for internal flow components where flow always stays subsonic or just sonic if choking occurs.

In order to help the solver find a subsonic solution to the problem, the equation actually implemented into the model keeps the same definition in the range of M from zero to unity and from M = 1 on defines a dummy parabolic curve with increasing slope from zero value at M = 1 so as to maintain continuity of the function and its derivative.

Component choking would be incurred whenever *F(M)* converges to solutions on the dummy curve. If that occurs the model currently flags a 'Fatal error' of choking condition in the component. A future development of the model will incorporate flow re-structuring, i.e. uncoupling of momentum and flow conservation equations such that flow is fixed at sonic conditions (critical mass flow) in a choking situation. This flow re-structuring, as aforementioned, may only be applicable if boundary conditions to the components in a line, other than fixed mass flow, are imposed. If mass flow is selected as a boundary condition, an error flag is unavoidable.

4.4 SELECTION OF TEARING VARIABLES

The main objective of the model generated is that, once it has been fully refined and has undergone a thorough validation process, it may become a useful tool of calculation of hydraulic resistance in gas flow networks.

One of the requirements that should be applied to this tool is that the user be able to produce output results with a minimum knowledge of the mathematical equations representing the physical system. The tool should be flexible enough to provide consistent results with a minimum of inputs and decisions made by the user. Even if the user is expert enough, with a good knowledge of the way the tool works, it is also a must that the amount of workload is reduced to a minimum.

The process of selection of suitable tearing variables to solve any network should be automatically done by the tool if possible. The selection made will obviously depend upon the set of boundary conditions selected. The model as now devised offers a selection of tearing variables by default which suits best the two usual different sets of boundary conditions considered, as indicated in the paragraphs above, so that the only input required from the user is selecting the set of boundary conditions and filling in their values. Tearing variables inisialisation is made within each component INIT code.

The typical tearing variables selected by the model for each component consist of: Mach number, M, at component ports and total pressure ratio, PRt (or altenatively total pressure differential, DPt) inside the components. Some other components have other variables selected too, e.g. flow ratio, Gr, in real flow dividers (see example in next chapter) but these have been kept to the absolute minimum.

5 COMPONENT EXAMPLES

The tool currently incorporates over 25 different types of components. Three of them are briefly described as typical examples of components including a description of their main features.

5.1 CIRCULAR BEND

Figure 2 shows the Smartsketch symbol associated.

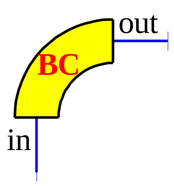

Figure 2: Circular bend Smartsketch symbol

Empirical data may be obtained from reference [4] about the pressure loss characteristics of a simple circular bend given through a pressure loss factor, K_{SG}, which is a function of the geometry and the Reynolds number. The component is assumed to be adiabatic so inlet and outlet temperatures will be equal.

Equation [4] allows to obtain the static pressure ratio across the bend as a function of inlet and outlet variables.

$$\Pi_s = \sqrt{\frac{TR_1}{TR_2} - \frac{F_{PH}^2}{TR_2}\cdot\left[K_{SG} - \frac{g+1}{g}\cdot\ln\left(\Pi_S\cdot\frac{TR_2}{TR_1}\right)\right]}$$

[4]

where

$$F_{PH} = \frac{G\cdot\sqrt{R\cdot T_t}}{A\cdot P} \quad (4.1)$$

is the hybrid flow parameter, and

$$TR = \frac{T_t}{T} = 1 + \frac{g-1}{2}\cdot M^2 \quad (4.2)$$

is the total to temperature ratio. The sub-indexes 1 and 2 indicate inlet and outlet positions respectively.

The transfer of pressure data between the inlet and outlet of a component, representing the momentum equation (which includes pressure loss) is standarised as a total pressure loss relationship, the only allowed in ports, so the static pressure ratio must be converted into total pressure ratio via equation [5], both related through the tearing variables of inlet and outlet M.

$$\Pi_s = \Pi_t\cdot\frac{\left(TR_1\right)^{\frac{g}{g-1}}}{\left(TR_2\right)^{\frac{g}{g-1}}} \qquad (5)$$

This method of calculation in the component will require the inlet and outlet Mach numbers and the total pressure ratio as tearing variables. The total pressure ratio variable will be marked as IMPL in the code to guide the solver.

5.2 REGULATING VALVE

Figure 3 shows the Smartsketch symbol associated.

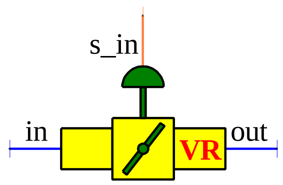

Figure 3: Regulating valve Smartsketch symbol

The text below, extracted from the respective EL file reflects the philosophy followed in the regulating valve to prevent problems with zero flow conditions.

```
CONTINUOUS

-- Links between valve inlet and outlet
out.G = in.G
in.G = A * in.Gac * fM(in.gamma, in_M)
IMPL (PRt) out.G = Aa * in.Gac * fM(in.gamma, Ma)
out.Pt = in.Pt * PRt
in.Tt = out.Tt

-- Geometric parameters
angle = s_in.signal
EXPAND (type == DN600 OR type == DN1000)
A = NP * PI * Dreg[type]** 2 / 4
EXPAND (type == Others)
A = NP * PI * D** 2 / 4

-- Calculation of Kv and XT (from manufacturer data)
Kv = poly(bound(angle, 0, 90), 7, K_600)
XT = poly(bound(angle, 0, 90), 5, XTf)

-- Calculation of Av and C1
Av = 28E-6 * Kv
C1 = 40 * sqrt (XT)

-- Reference Area
Aa = NP * 24.4E-3 * Av * C1 * in_dtf**((in.gamma + 1) / 2 / (1 -
in.gamma))

-- Total pressure drop
Pad = 1 + (PRt - 1) / (1.2e-3 * spow2(C1) * in_dtf**((in.gamma +
1) / (1 - in.gamma)))
Ma = ssqrt(2 / (in.gamma - 1) * (Pad**((1 - in.gamma) / in.gamma)
- 1))

-- Calculation of fluid data
f_data (in.gamma, in.Pt, in.Tt, in_M, in_P, in_T, in_rho, in_V,
in_dtf)
```

It may be seen that once the total pressure ratio is known (or guessed), together with a guess made of inlet M, the model calculates the static to total pressure ratio at the valve orifice inlet, P_{ad}, and then the valve orifice inlet Mach number, M_a, which is used by the outlet flow equation to match the inlet flow. However, if the valve orifice area is zero (closed valve), flow will be set to null independent on the value of M_a generated.

5.3 DIVIDING FLOW JUNCTION

Figure 4 shows the Smartsketch symbol associated.

This component provides pressure loss characteristics in the two legs of a dividing 180 degree junction. This type of components together with the flow combiners allow the construction of internal flow networks with loss characteristics in the junctions compared to an ideal junction which is also allowed by the component port features which make use of EcosimPro power.

It includes the flow conservation, momentum and energy equations which link the three connections each other. This component does not need any guess on pressure ratio (or pressure differential), i.e. PRt (or DPt) are not marked as IMPL, as these can be calculated directly by guessing common inlet Mach number (this is similar to other components which can calculate pressure loss characteristics without requiring any input guess from the outlet section. More than required tearing variables usually yield convergence problems.

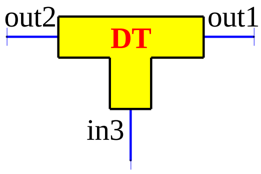

Figure 4: 180 degree divider Smartsketch symbol

The text below illustrate the structure of this component, where K_{31} and K_{32} correspond to the pressure loss characteristics obtained from bibiliography, which are a function of geometry, inlet Reynolds number and flow split ratio. The flow split ratio G_1/G_3 is introduced as a guess and used to match port pressures in the new branch introduced into the network.

```
CONTINUOUS

-- Links between the inlet pipe and the outlet pipes
out1.G = in3.G * Gr
in3.G = A3 * in3.Gac * fM(in3.gamma, in3_M)
in3.G = out1.G + out2.G
in3.Pt = out1.Pt + DPt1
in3.Pt = out2.Pt + DPt2
in3.Tt = out1.Tt
in3.Tt = out2.Tt

-- Geometric parameters
A1 = NP * PI * D1**2 / 4
A2 = NP * PI * D2**2 / 4
A3 = NP * PI * D3**2 / 4
Ar = A1 / A3

-- Adimensional/semi-adimensional parameters and numbers
Re = bound(4 * in3.G / (PI * D3 * Dv(in3_T)), 1, Inf)

-- Calculation of total pressure loss in both legs
DPt1 = K31 * in3.gamma * in3_P * spow2(in3_M) / 2
DPt2 = K32 * in3.gamma * in3_P * spow2(in3_M) / 2

-- Calculation of fluid data
f_data (in3.gamma, in3.Pt, in3.Tt, in3_M, in3_P, in3_T, in3_rho,
in3_V, in3_dtf)

END COMPONENT
```

6 CONCLUSIONS

EcosimPro has proven to provide excellent performance in simulation of relatively complex internal steady flow flow networks, providing a friendly interface to the user. The use of the variable time as an additional parameter to carry out parametric analyses in steady flow simulations and the possibility of building experiments with different types of boundary conditions are some of the features that makes EcosimPro a flexible tool and a very good choice for this task.

Figure 5 shows a typical example of a simple internal flow network built with EcosimPro.

References

[1] The Dynamics and Thermodynamics of Compressible Fluid Flow. Volume I. A.H. Shapiro. The Ronald Press Company. New York

[2] Internal Flow Systems. D.S. Miller. BHR Group Limited.

[3] Handbook of Heat Transfer. W.M. Rohsenow, J.P. Harnett, Y.I. Cho. 3rd edition. Mc Graw Hill.

[4] E.S.D.U. Fluid Mechanics, Internal Flow. Volumes 1 to 4.

[5] Handbook of Hydraulic Resistance. I.E. Idelchik. 3rd edition. Begel House.

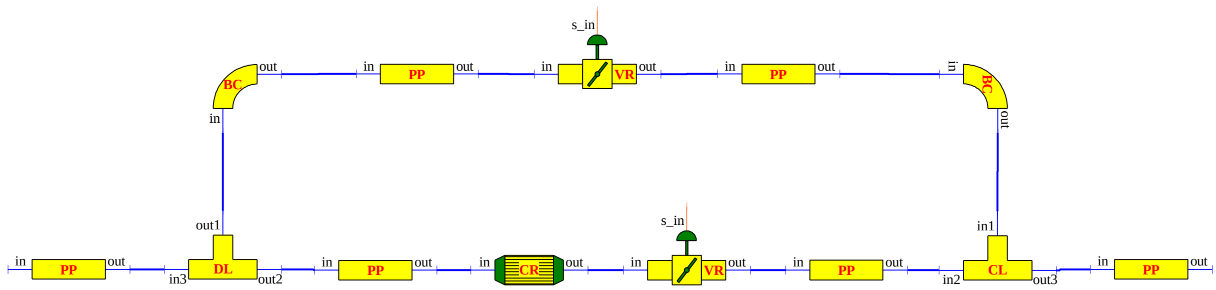

Figure 5: Smartsketch layout of a simple network showing a flow temperature control via an intercooler with flow split control through butterfly valves