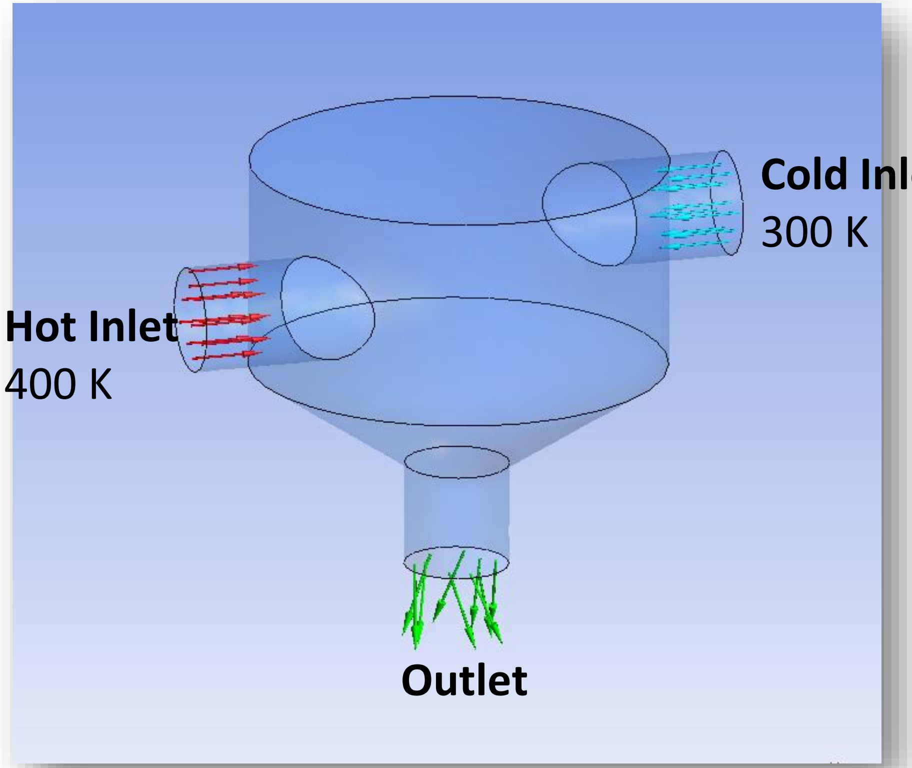
Optimum Design Course
Fall 2016

Ansys Tutorial 8

Mixing Tank

Problem Description

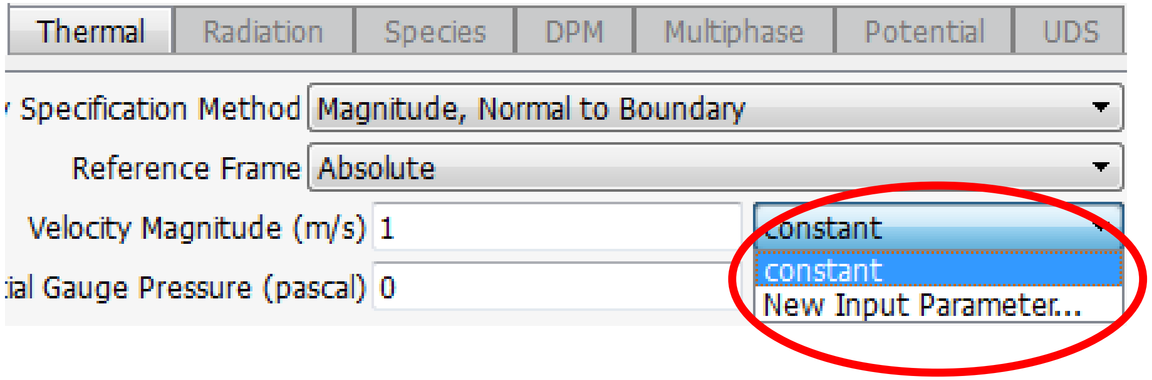
- This workshop looks deeper into the options available for DOEs, Response Surfaces and Optimization, as well as exposes you to creating parameters in FLUENT and CFD-Post
- The problem to be analyzed is a static mixer where hot and cold fluid entering at variable velocities mix. The objective of this analysis is to find inlet velocities which minimize pressure loss from the cold inlet to the outlet and minimize the temperature spread at the outlet
- Input
  - Hot inlet velocity
  - Cold inlet velocity
- Output
  - Pressure loss
  - Temperature spread

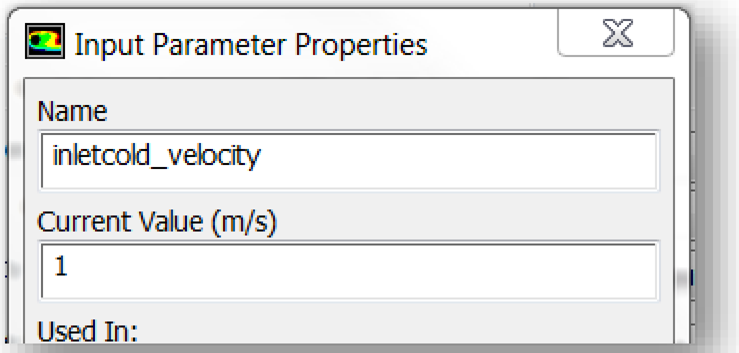


Parameterize Project

- **Start Workbench and open *mixingTank_initial.wbpj*.**

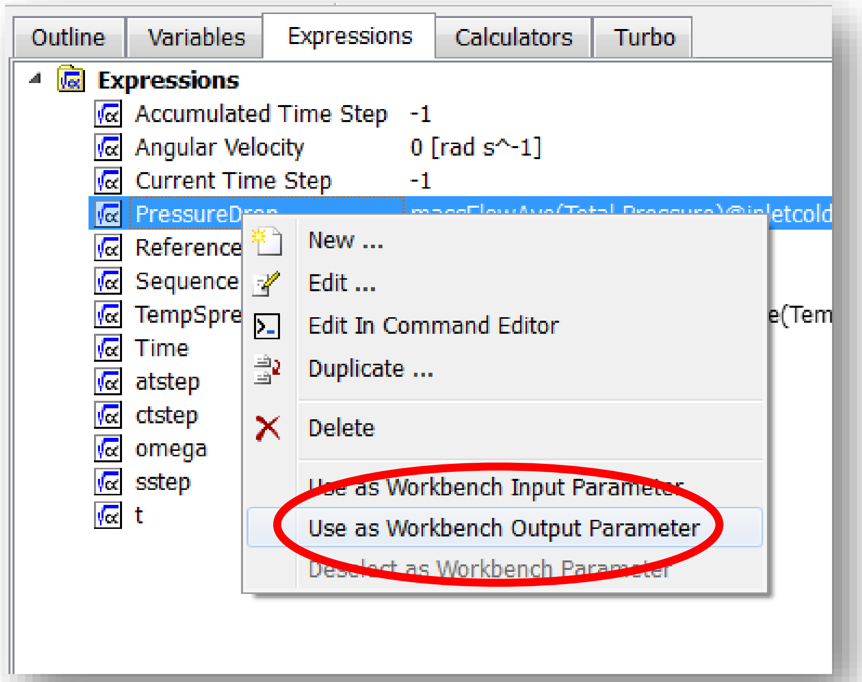
- **Parameterize the boundary conditions in FLUENT:**
  - Double-click *Setup*. Click OK on pop-up windows
  - In the *Tree Outline*, select *Boundary Conditions* and in the Boundary Conditions list, click *inletcold* and select *Edit*
  - In the *Velocity Inlet* window, click the *Constant* drop down menu next to *Velocity Magnitude* and select *New Input Parameter*. Set *Name* to *inletcold_velocity* and *Current Value* to 1. Click *OK* to close both pop-up windows
  - Repeat for *inlethot at 2.5m/s* (use *inlethot_velocity* as the parameter name)
  - Close *FLUENT*





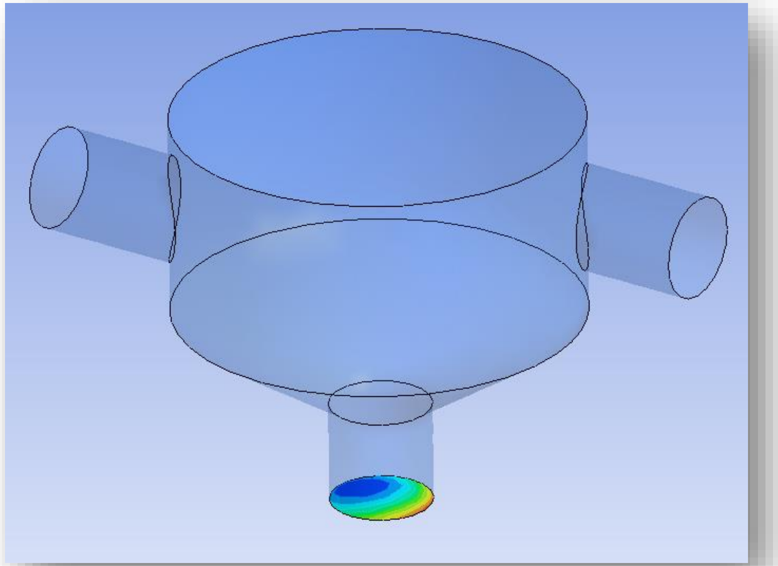
Parameterize Project

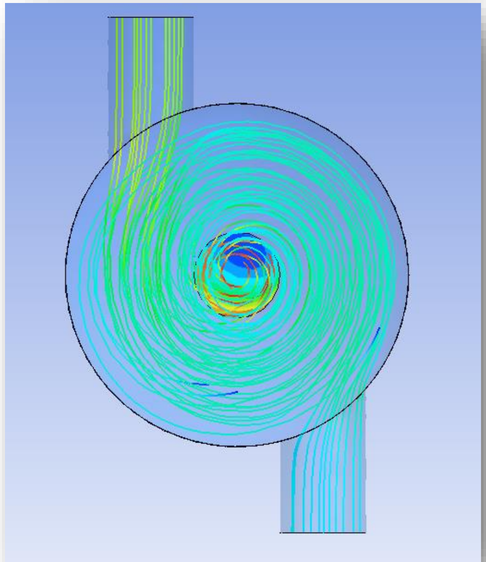
- **Parameterize the output parameters in CFD-Post:**
  - **Double-click *Results***
  - **Parameterize the Pressure Drop: Click the *Expressions* tab > right-click *PressureDrop* > select *Use as Workbench Output Parameter***
  - **Parameterize the Temperature Spread: Repeat for the expression called *TempSpread***



Review Model

- **Review the results**
  - **Observe the temperature distribution at the outlet:** Click *Insert* > click *Contour* > click *OK.* In the *Details* view, set *Locations* to *pressureoutlet* > set *Variable* to *Temperature* > set *Range* to *Local* > click *Apply*. Notice that the temperature distribution is non-uniform but the temperature varies by less than 2 Kelvin
  - **Observe streamline from the inlets:** Click *Insert* > click *Streamline* > click *OK.* In the *Details* view, set *Start From* to *inletcold* and *inlethot* (select two locations by clicking '…') > click *Apply*. Notice that due to the off-alignment of the inlets the fluid swirls through the tank
- **Close CFD-Post**

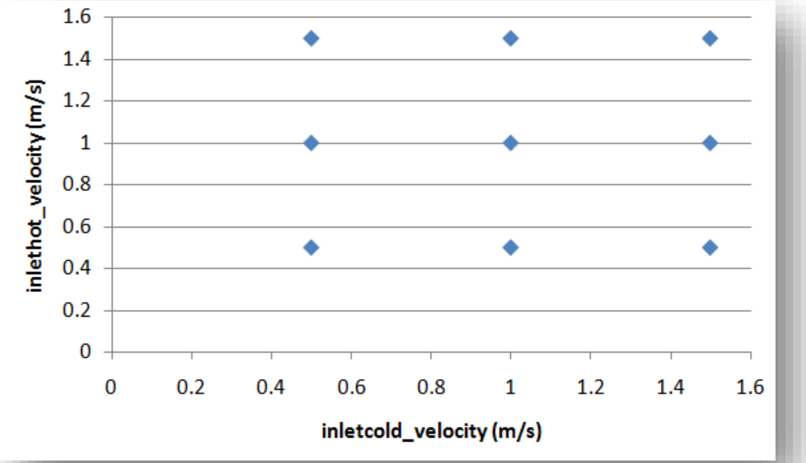




Conduct a DOE

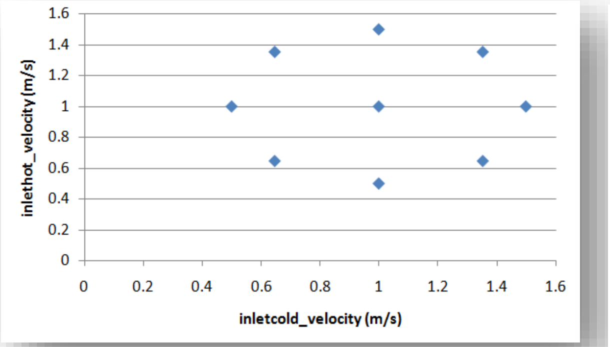
- Drag and drop a Response Surface *Optimization* system onto the *Project Schematic* > double-click *Design of Experiments*

- **Explore the different DOE types**
  - **Select the CCD DOE:** In *Outline of Schematic*, click *Design of Experiments (A2)* > in the Properties of Outline set *Design of Experiments Type* to *Central Composite Design*
  - **Specify the ranges for each input parameter:** In *Outline of Schematic*, click *P1 – inletcold_velocity* (A5) > in the *Properties of Outline* set *Lower Bound* to *0.5* > set *Upper Bound* to *1.5*. Repeat for *P2 – inlethot_velocity*
  - **Preview the DOE:** In the top toolbar click *Preview*. DesignXplorer will create design points based on the DOE type, the number of input parameters, and the parameter ranges. The design point selections for this DOE are illustrated in the following image. (This figure is not displayed in DX, it was created in a 3rd party application)

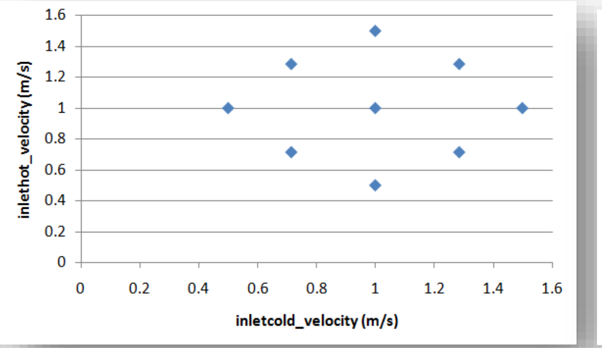


Conduct a DOE

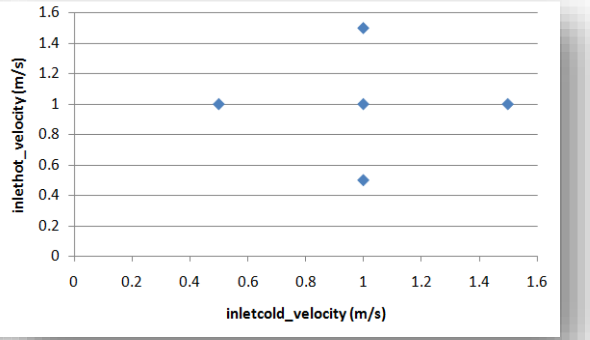
- Repeat the process outlined on the previous slide for different DOEs. Considering time restriction, do not Update the DOE since this will solve each design point. (The figures below are not from DX, they were created in a 3rd party application)
- To save time, instead of solving the design points, load the project where the CCD DOE has already been conducted. Click *File* > click *Open* > click *No* for any pop-up windows > select *mixingTank_CCD.wbpj* (Note the min/max velocities are different in the new file)



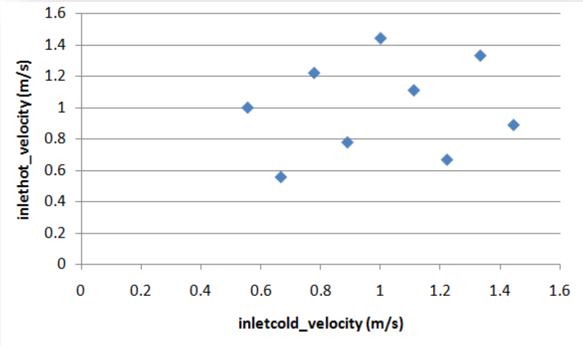
CCD – Rotatable (standard)



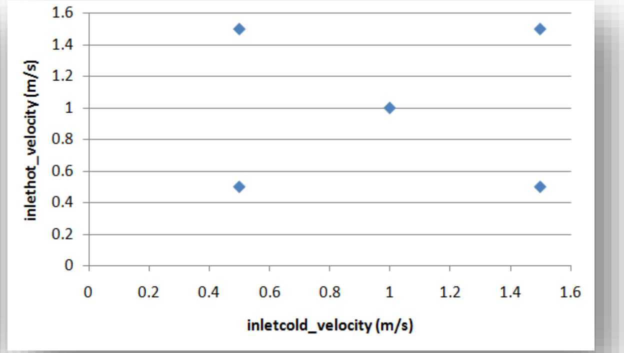
CCD – G-Optimality



Sparse Grid



Optimal Space Filling

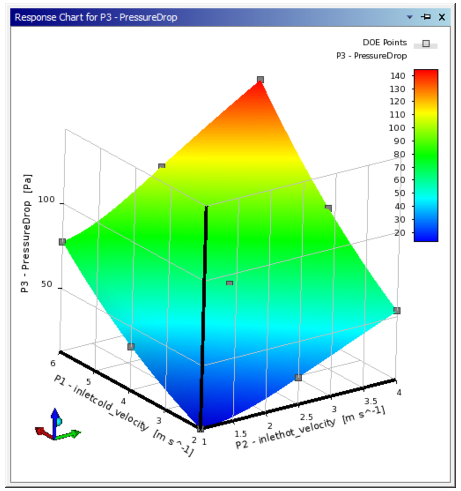


Box-Behnken Design

Explore the Response Surfaces

- **Review the Full 2nd Order Polynomials response surface**
  - **Generate the Full 2nd Order Polynomials Response Surface:**
  - **From the Workbench project page, double click on ‘Response Surface’, then in the outline of schematic click on Response Surface (A2). In the Properties window, ensure that *Response Surface Type* is set to *Standard Response Surface – Full 2nd Order Polynomials***
  - **In the toolbar, click Update > If a window pops up click Yes**

  - Observe Pressure Drop as a function of the inlet velocities:
  - In *Outline of Schematic* under *Response Points*, click *Response (A16)*
  - In *Properties of Outline* set:
    - *Mode: 3D*
    - *Tick “Show Design Points”*
    - *X Axis: inletcold_velocity*
    - *Y Axis: inlethot_velocity*
    - *Z Axis: PressureDrop*



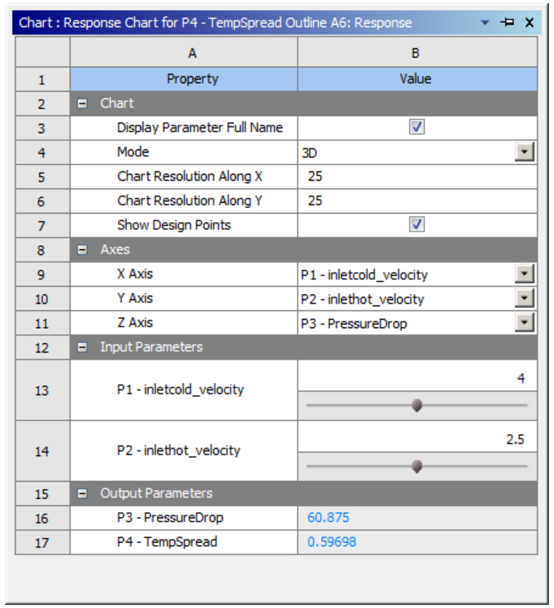


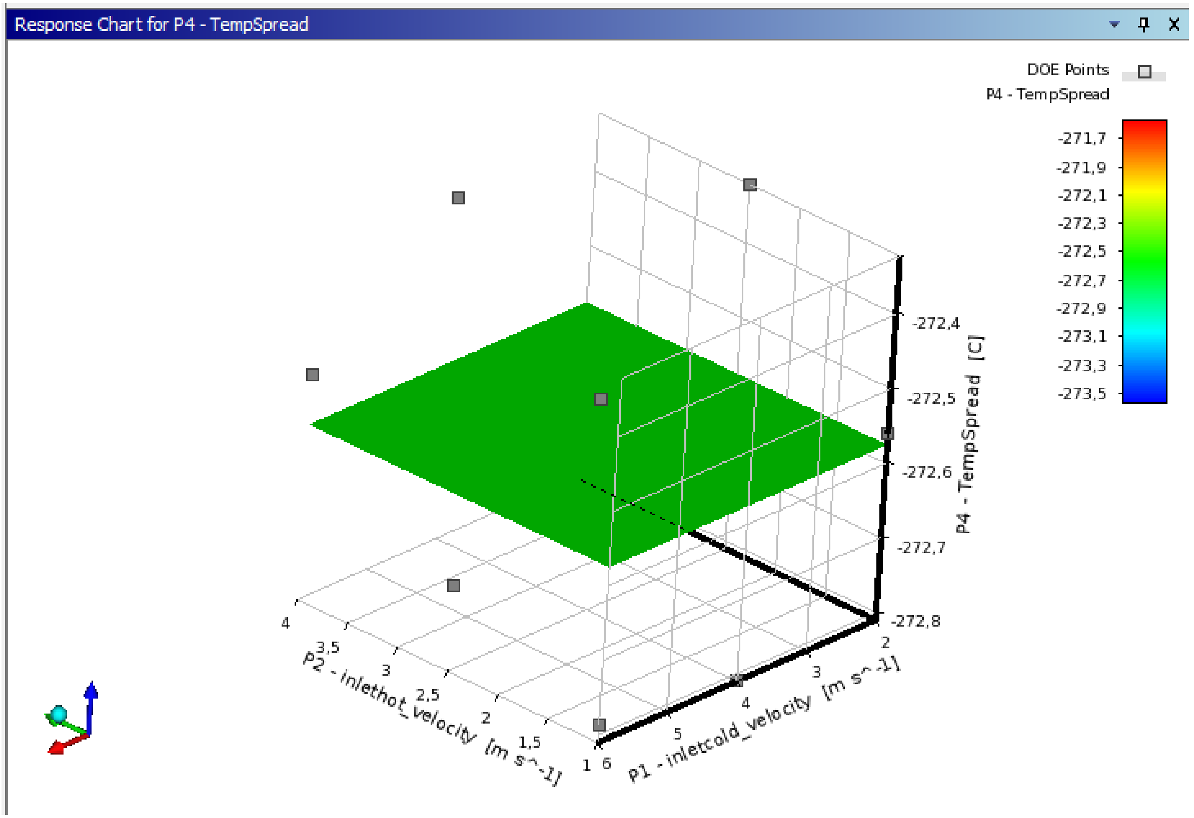
Chart : Response Chart for P4 - TempSpread Outline A6: Response

	A	B
1	Property	Value
2	Chart	
3	Display Parameter Full Name	☑
4	Mode	3D
5	Chart Resolution Along X	25
6	Chart Resolution Along Y	25
7	Show Design Points	☑
8	Axes	
9	X Axis	P1 - inletcold_velocity
10	Y Axis	P2 - inlethot_velocity
11	Z Axis	P3 - PressureDrop
12	Input Parameters	
13	P1 - inletcold_velocity	4
14	P2 - inlethot_velocity	2.5
15	Output Parameters	
16	P3 - PressureDrop	60.875
17	P4 - TempSpread	0.59698

Notice that the response surface cuts through all design points, an indicator that the response surface is well suited for this parameter. The Full 2nd Order Polynomials surface works well for flat responses

Explore the Response Surfaces

- **Observe Temperature Spread as a function of the inlet velocities:**
- **In *Properties of Outline* change the *Z Axis* to *TempSpread*. Notice that the response surface poorly represents the design points, an indicator that the response surface is poorly suited for this problem. Full 2nd Order Polynomials response surfaces do not represent non-linear responses well**



Explore the Response Surfaces

– **Review the Goodness of Fit:** In *Outline of Schematic* under *Metrics*, click *Goodness of Fit*. **In Table of Outline, notice that TempSpread has poor goodness of fit metrics which agrees with what was observed in the previous step**

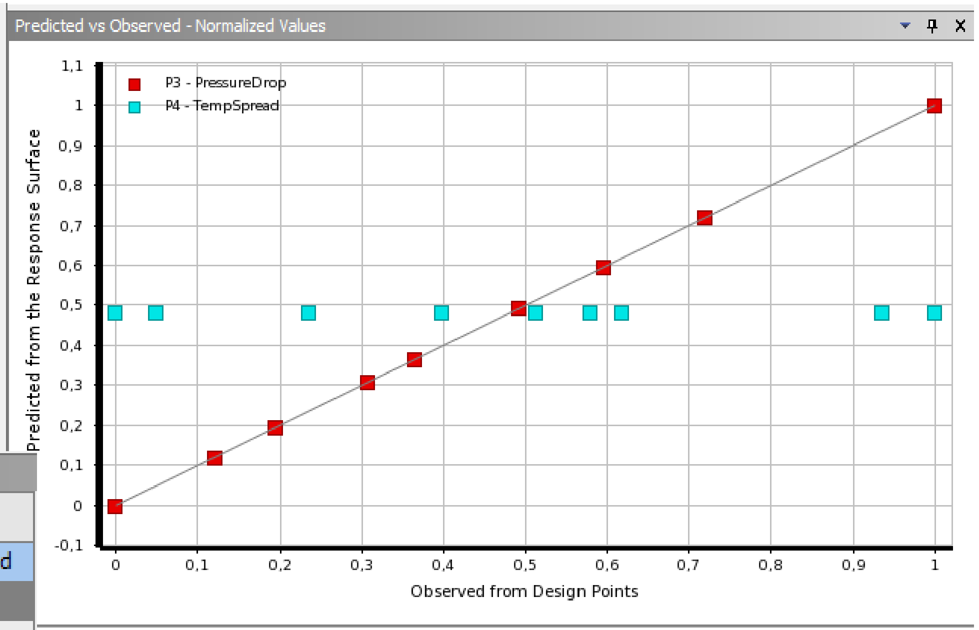
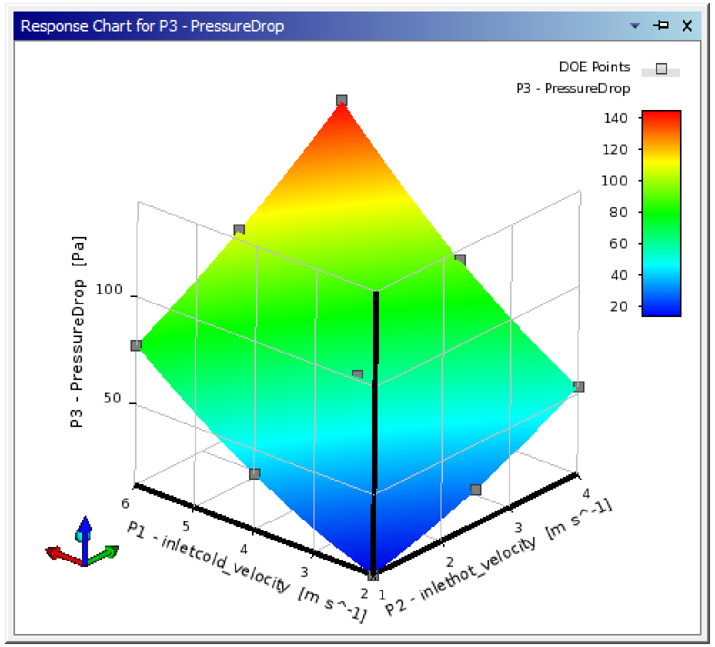


Table of Schematic B3: Response Surface

	A	B	C
1		P3 - PressureDrop	P4 - TempSpread
2	Coefficient of Determination (Best Value = 1)		
3	Learning Points	1	0
4	Root Mean Square Error (Best Value = 0)		
5	Learning Points	1.0366E-06	0.16207
6	Relative Maximum Absolute Error (Best Value = 0%)		
7	Learning Points	0	147.48
8	Relative Average Absolute Error (Best Value = 0%)		
9	Learning Points	0	78.375

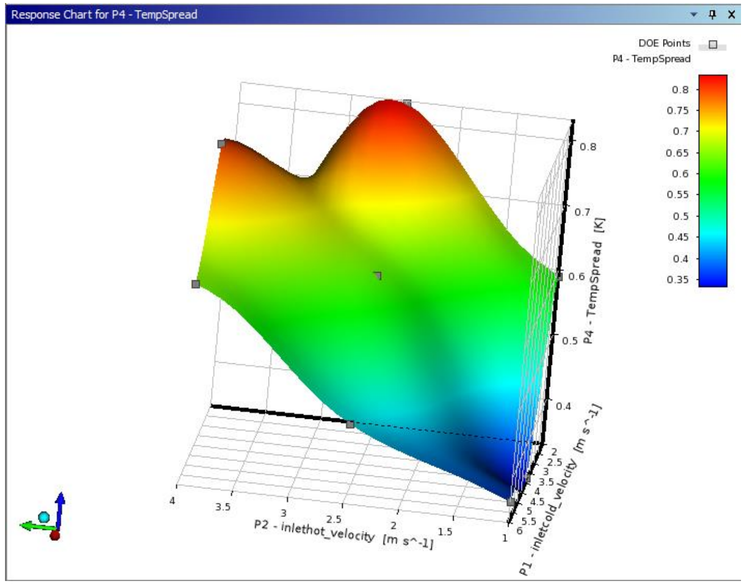
Explore the Response Surfaces

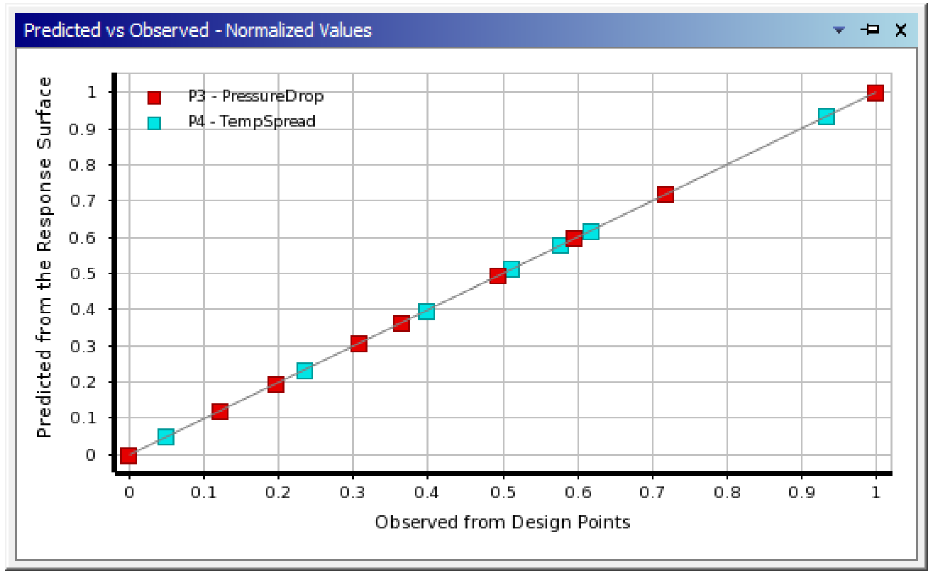
- **Review the Genetic Aggregation response surface**
  - **Generate the Genetic Aggregation Response Surface:** In the *Outline of Schematic* click *Response Surface* (A2) > set the *Response Surface Type* to its default: *Genetic Aggregation* > in the toolbar, click Update
  - **Observe Pressure Drop as a function of the inlet velocities:** Try accomplishing this without instructions. Otherwise, follow same procedure as described on the previous slide. Notice that the response surface cuts through all design points. The Genetic Aggregation response surface always cuts through all of the design points so this alone is not a measure of the goodness of fit



Explore the Response Surfaces

- **Observe Temperature Spread as a function of the inlet velocities (same workflow as before):** **The Genetic Aggregation response surface cuts through all design points. If the response surface does not pass through all points, try increasing the Number of Points on X and Y**
- **Review the Goodness of Fit:** **Since the Genetic Aggregation response surface cuts through all design points, the goodness of fit metrics are always perfect. This, however, does not imply that the response surface properly predicts the response away from the design points. One way to check the goodness of fit for Genetic Aggregation response surfaces is to look at the Cross-Validation on Learning Points and to compare it with one or more verification points. Verification points are additional design points that are not used when generating the response surface, consequently the response surface will not necessarily cut through the verification points (the same applies to the Sparse Grid and Kriging response surfaces)**





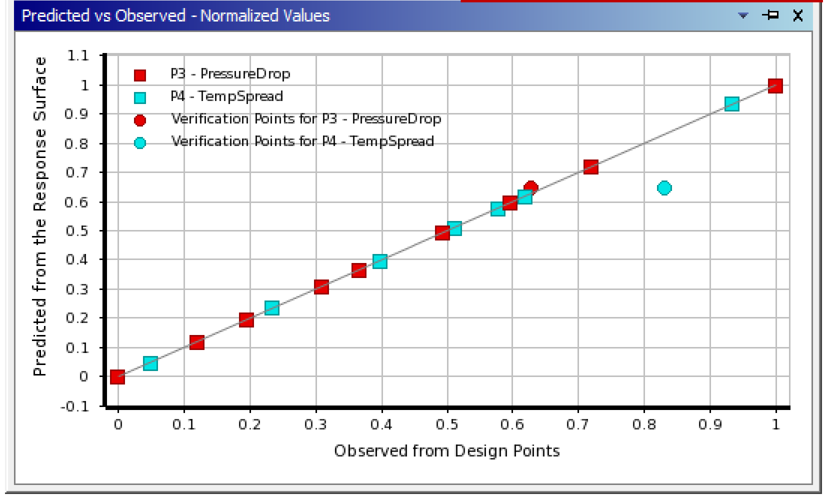
Explore the Response Surfaces

- **Review the Genetic Aggregation response surface**
  - **Look at the Cross-Validation on Learning Points:** **the Cross Validation shows that the results are bad away from the calculated points**
  - **Add Verification Points:** **In the *Outline of Schematic*, click *Response Surface (A2)* > in the *Properties of Outline* enable *Generate Verification Points* > retain the default N*umber of Verification Points* of *1*. *Click 'Update'*.** **One additional condition will be solved for. This may take a few minutes. In practice, it is suggested that you create several (e.g. 3) verification points to gain a deeper understand of the goodness of fit**
  - **Review the Goodness of Fit:** **In the *Outline of Schematic, click Goodness of Fit.*** **The goodness of fit metrics are displayed at the bottom of the *Table of Outline.* Notice that the goodness of fit metrics are not perfect and are actually quite poor specially for TempSpread. This indicates that the response surface is not ideal and that alternate approaches should be investigated**

Note: The Genetic Aggregation generally does not require verification points because the Cross-Validation gives very often equivalent results as you can see in this example

Table of Schematic B3: Response Surface

	A	B	C
1		P3 - PressureDrop	P4 - TempSpread
2	Coefficient of Determination (Best Value = 1)		
3	Learning Points	1	1
4	Cross-Validation on Learning Points	1	0
5	Root Mean Square Error (Best Value = 0)		
6	Learning Points	9.7659E-06	6.3379E-09
7	Verification Points	2.7804	0.11812
8	Cross-Validation on Learning Points	2.7182E-05	0.19843
9	Relative Maximum Absolute Error (Best Value = 0%)		
10	Learning Points	0	0
11	Verification Points	6.9518	69.117
12	Cross-Validation on Learning Points	0.00010441	274.67
13	Relative Average Absolute Error (Best Value = 0%)		
14	Learning Points	0	0
15	Verification Points	6.9518	69.117
16	Cross-Validation on Learning Points	6.2473E-05	86.058



Explore the Response Surfaces

- ## Genetic Aggregation response surface refinement

  – **Add Refinement Points: One benefit of the Genetic Aggregation response surface is that it allows for automatic refinement. Refinement points are additional design points that are used to improve the response surface. Automatic refinement allows for refinement points to try achieve the Maximum Predicted Error that you enter yourself for the chosen output parameter**

  – **In the *Outline of Schematic*, click *the Output Parameters* > in the *Table of Schematic look at the minimum and maximum calculated values and at the maximum predicted error***

Table of Schematic B3: Response Surface: Tolerances

	A	B	C	D	E	F
1	Name	Calculated Minimum	Calculated Maximum	Maximum Predicted Error	Refinement	Tolerance
2	P3 - PressureDrop (Pa)	12.816	144.26	2.0482E-05	☐	
3	P4 - TempSpread (K)	0.33154	0.83217	0.17052	☐	

  – ***As you can see, the Pressure Drop does not need refinement because the predicted error is very low comparing to the minimum and maximum values. This is not the case for the Temperature Spread***

  – **Click on refinement for the P4 – TempSpread output and enter a tolerance of 0.05** ***This will continue adding refinement points until the error reaches 0.05. In this case, only 3 refinement points are added in the Properties of outline of Response Surface due to time constraints***

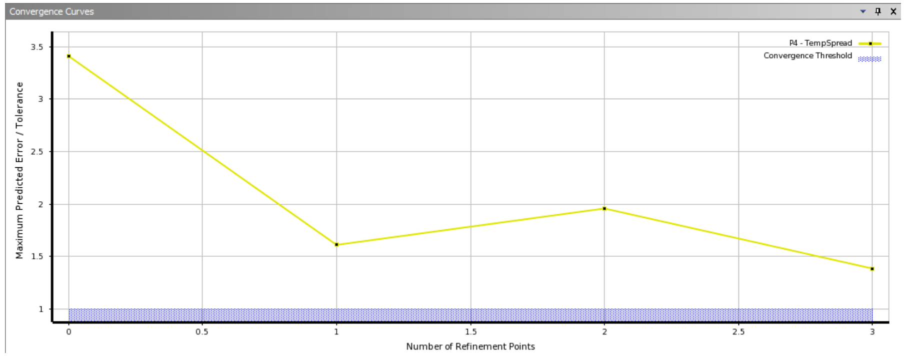
  – **update**

6	⊟ Meta Model	
7	Response Surface Type	Genetic Aggregation
8	⊟ Refinement	
9	Maximum Number of Refinement Points	3
10	Number of Refinement Points	0

	A	B	C	D	E	F
1		...m	Calculated Maximum	Maximum Predicted Error	Refinement	Tolerance
2			144.26	2.0482E-05	☐	
3	P4 - TempSpread (K)	0.33154	0.83217	0.17052	☑	0.05

Explore the Response Surfaces

- **Review the response surface and the maximum predicted error**
  - **Review the Convergence Curves: In the lower right frame you should see convergence curves. The convergence curve indicates the current predicted relative error as a function of number of refinement points. Notice that the pressure drop curve is within the acceptable error but the temperature spread curve is far outside the range of acceptable error**



  - **Review the maximum predicted error and the Response Surface: Check the response surface, the goodness of fit and notice that the calculated maximum predicted error is 0.069216 greater than the tolerance**

Table of Schematic B3: Response Surface: Tolerances

	A	B	C	D	E	F
1	Name	Calculated Minimum	Calculated Maximum	Maximum Predicted Error	Refinement	Tolerance
2	P3 - PressureDrop (Pa)	12.808	144.26	2.7415	☐	
3	P4 - TempSpread (K)	0.3261	0.8956	0.069216	☑	0.05

Note: In this model, even if we add 20 refinement points, the predicted error won't get better. You should always pay attention to the choice of your output parameter, to the mesh quality and the overall model. In this example the mesh was not fine enough due to time constraint

Explore the Response Surfaces

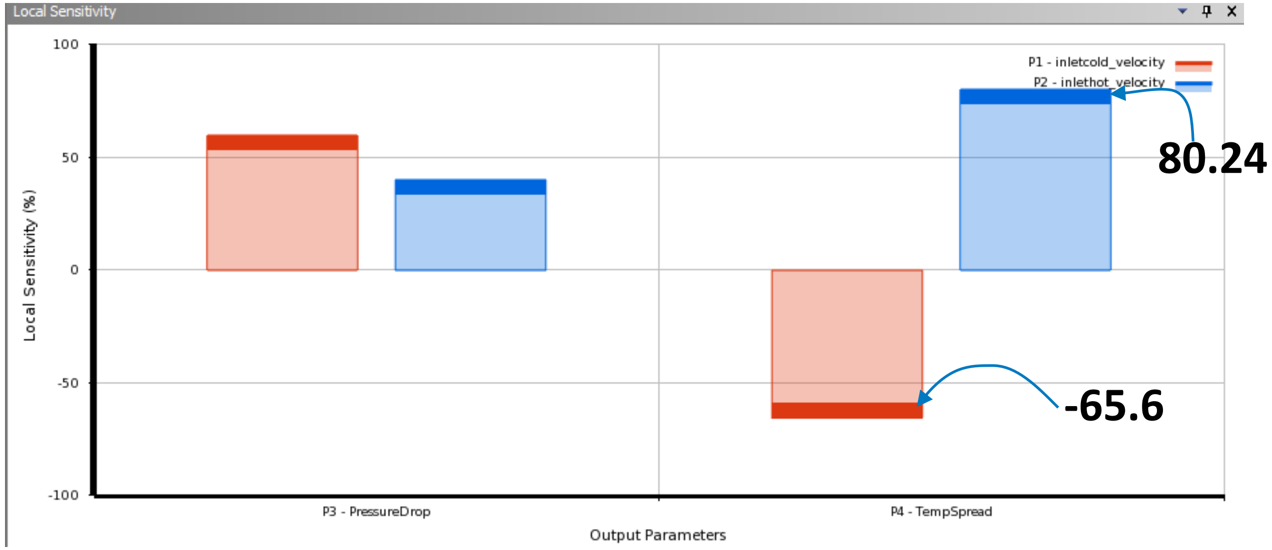
- **Review the local sensitivity**
  - **In the *Outline of Schematic*, *click Local Sensitivity* > in the *Properties of Outline*, ensure that *inlethot_velocity* is set to 2.5 and *inletcold_velocity* is set to 4**
  - **The local sensitivity of temperature spread to the two input parameters is shown in the image below. The local sensitivity is calculated according to the equation :**

$$\frac{(Output_{\max} - Output_{\min})_{local}}{(Output_{\max} - Output_{\min})_{global}}$$

where $(Output_{\max} - Output_{\min})_{local}$ *is calculated when one input varies and* $(Output_{\max} - Output_{\min})_{global}$ *is calculated when all the inputs vary*

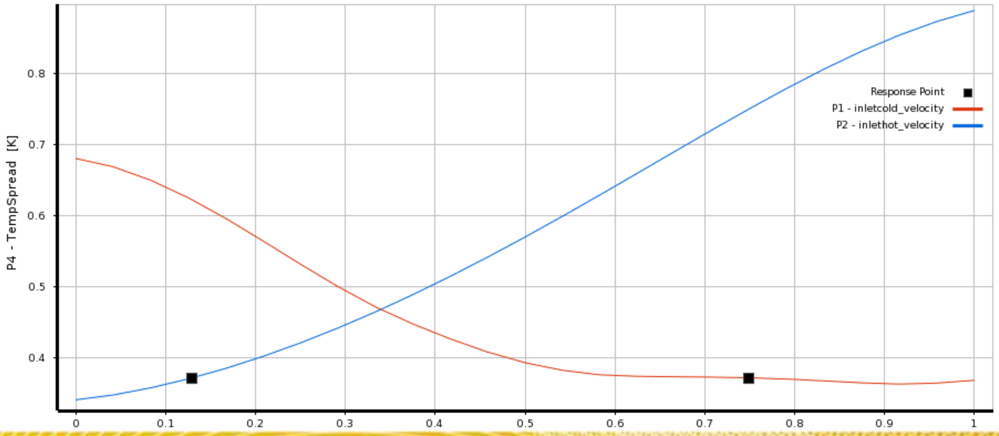
So, if inletcold_velocity is held at 4 m/s, the sensitivity of the temperature spread to inlethot_velocity is 80.24%

Similarly, when inlethot_velocity is held at 2.5 m/s, the sensitivity of the temperature spread to inletcold_velocity is -65.6%



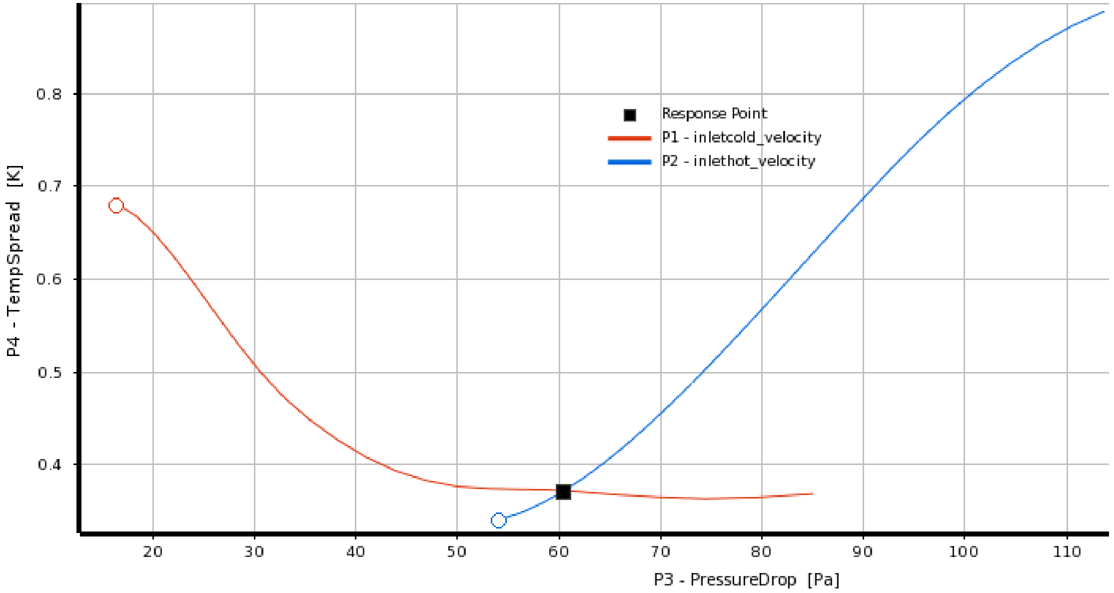
Explore the Response Surfaces

- **Review the local sensitivity**
  - In the *Outline of Schematic, click Local Sensitivity Curves* > in the *Properties of Outline*, set Y Axis to *TempSpread*, *inletcold_velocity* to 5 and *inlethot_velocity* to 1.39
  - The local sensitivity curves of temperature spread to the two input parameters is shown
  - This shows that:
    - When inlethot_velocity is held at 1.39 m/s, variations in inletcold_velocity will result in temperature spread values ranging from 0.36-0.68 (red line)
    - When inletcold_velocity is at it's smallest value the temperature spread will be 0.68 and when inletcold_velocity is at it's largest value the temperature spread will be 0.36
    - Similarly, when inletcold_velocity is held at 5 m/s, variations in inlethot_velocity will cause result in temperature spread values ranging from 0.34-0.88 (blue line). Thus, at these conditions, the temperature spread is slightly more sensitive to inlethot_velocity



Explore the Response Surfaces

- **Review the local sensitivity**
  - **Review the local sensitivity curves:** In the ***Properties of Outline***, set the ***x-axis*** to PressureDrop and the ***y-axis*** to TempSpread
  - This shows that when inlethot_velocity is held at 1.39 [m/s], the output parameter values will vary according to the red line. At the lowest value of inletcold_velocity (as indicated by the circle) the temperature spread is 0.68 [K] and the pressure drop is 16.5 [Pa]. At the highest value of inletcold_velocity the temperature spread is 0.36 [K] and pressure drop is 84 [Pa]
  - When inletcold_velocity is held at 5 m/s, the blue line shows the relationship between output parameters. Since the blue line is more vertical than horizontal it can be concluded that when inletcold_velocity is held at 5 m/s, the temperature spread is more sensitive to inlethot_velocity than the pressure drop



- **Switch to the Project page**

Optimize the design

- **Explore the *Screening* optimization method**
  - In the *Project Schematic*, double-click *Optimization [B4]*
  - In the *Outline of Schematic*, click Optimization (A2) > In the *Properties of Outline* ensure *Optimization Method* is set to *Screening*. Set *Number of samples* to *1000*
  - **Specify the region of the design space to investigate:** In the *Table of Schematic,* set inlethot_velocity lower bound to 1.5 and upper bound to 3.5. Retain the defaults for inletcold_velocity
  - **Specify the optimization objectives:** Click Objectives and Constraints. In the Table of Schematic, set the TempSpread objective and PressureDrop objective to minimize. Set PressureDrop objective to have a higher importance

Table of Schematic B4: Optimization

	A	B	C	D	E	F	G
1	Name	Parameter	Objective		Constraint		
2			Type	Target	Type	Lower Bound	Upper Bound
3	Minimize P4	P4 - TempSpread	Minimize		No Constraint		
4	Minimize P3	P3 - PressureDrop	Minimize		No Constraint		
*		Select a Parameter					

Properties of Outline A9: Minimize P3

	A	B
1	Property	Value
2	General	
3	Parameter	P3 - PressureDrop
4	Objective Name	Minimize P3
5	Objective	
6	Type	Minimize
7	Constraint	
8	Type	No Constraint
9	Decision Support Process	
10	Objective Importance	Higher
11	Parameter Details	
12	Units	Pa
13	Calculated Minimum	12.816
14	Calculated Maximum	144.26

Optimize the design

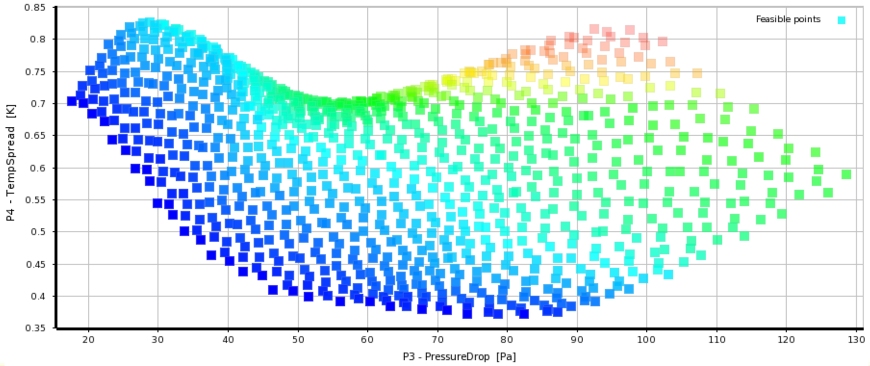
- **Explore the *Screening* optimization method**
  - **Perform optimization: Click Update. The screening method will create 1000 sample points based on the response surface, compare them, and select the three samples which best satisfy the objectives**
  - **Review the three candidates**
  - **Verify candidate 1: The sample data is based on the response surface, so it is important to verify that the system performs as expected under these conditions. Right-click Candidate 1 and select Verify by Design Point Update. This may take a few minutes. Once complete, the verification results will be displayed in the Table of Schematic. Compare the response optimization prediction with the verification results. If there is a substantial difference, the accuracy of the response surface should be assessed and refined if needed**

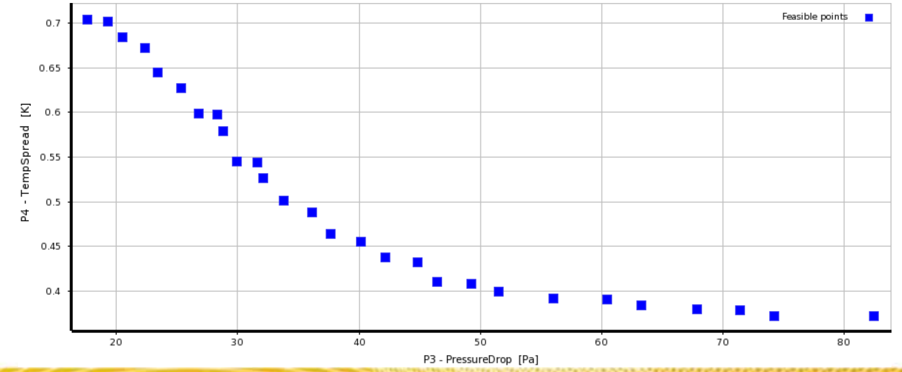
8	Candidate Points				
9		Candidate Point 1	Candidate Point 1 (verified)	Candidate Point 2	Candidate Point 3
10	P1 - inletcold_velocity (m s^-1)	3.538		4.05	4.434
11	P2 - inlethot_velocity (m s^-1)	1.5127		1.503	1.5498
12	P3 - PressureDrop (Pa)	★★ 37.717	★★ 36.876	★ 46.449	★ 54.392
13	P4 - TempSpread (K)	★★ 0.46453	★ 0.48686	★★ 0.41081	★★ 0.40185

The computed pressure drop of 36.8 Pa is close to the predicted 37.7 Pa

Optimize the design

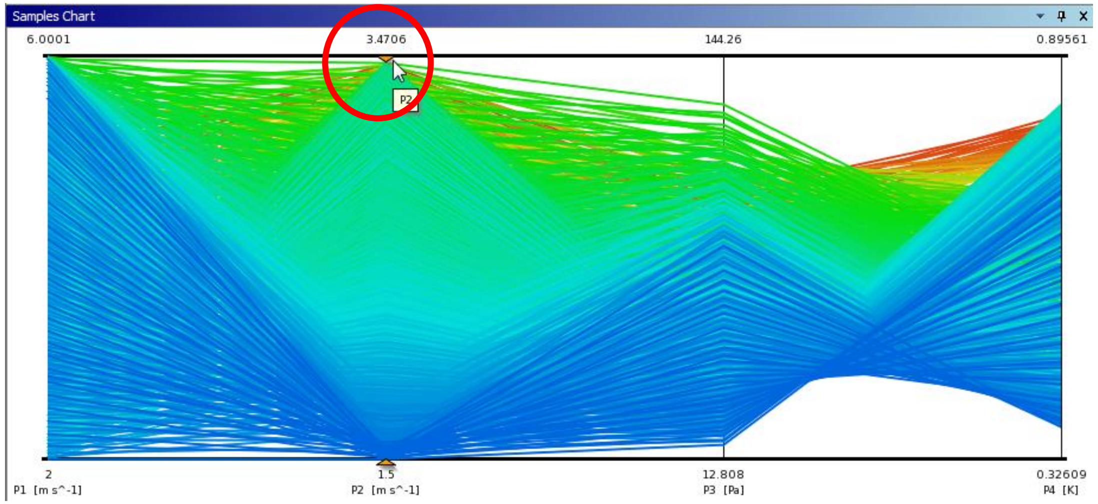
- **Explore the *Screening* optimization method**
  - **Review the tradeoff chart: In the *Outline of Schematic*, click *Tradeoff*. The tradeoff chart shows the output parameter values for each sample. Hovering the cursor above any point will display the input parameters used**
  - **Explore the Pareto fronts: Notice that the samples are in different colors. The color indicates the Pareto front, with blue being the lowest Pareto front and red being the highest Pareto front. The more blue a sample is, the better it satisfies the objectives**
  - **Notice that there are competing objectives. The sample with the lowest temperature spread has a very high pressure drop. Similarly, the sample with the lowest pressure drop has a very high temperature spread. The Screening optimization picked three candidates but those are not necessarily the top candidates**
  - **Review the first Pareto front: In the *Properties of Outline*, set *Number of Pareto Fronts to Show* to *1*. The first Pareto front contains the top samples**
  - **Create a design point for the sample which you think is the best: Right-click a sample from the first Pareto front select insert as Design Point. This will add a design point to the Parameter set for this condition**

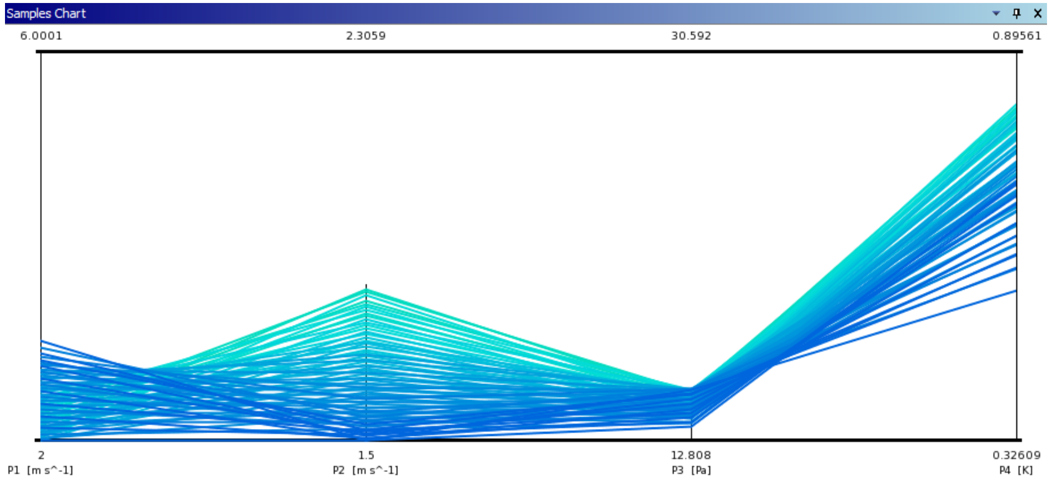




Optimize the design

- **Explore the *Screening* optimization method**
  - **Review the Samples chart:** In the *Outline of Schematic*, click Samples, then in the properties window, for Mode (B4) select ‘Candidates’. The samples chart shows the parameter combinations for each sample. The green lines correspond to the candidates
  - **Narrow the parameter ranges:** In the Samples chart, hover the cursor just below the 3.5 in the P2 column and drag the triangle down. In the image below, doing so would hide all samples with P1 greater then 2.3 (look at the top row of the graph to see the value you are sliding the marker down to). The samples chart allows you to easily find the desired samples by allowing you to dynamically control the allowable range for each parameter





  - Adjusting the parameter ranges can help determine an area to focus further optimization studies on. Drag P3 down to approximately 30K, and P4 to 0.62Pa. This range for P1 and P2 will be used in the following steps

Optimize the design

- **Explore the *MOGA* optimization method**
  - **Select the MOGA (Multi-Objective Generic Algorithm) method:** In the *Outline of Schematic*, click Optimization (A2) > In the *Properties of Outline* set *Optimization Method* to *MOGA.* Review the other options related to MOGA
  - **Narrow the optimization domain**: Select each of the 2 inlet velocities in the *Outline of Schematic*, then in the *Table of Schematic*, set the *lower bound* of P1 (inletcold_velocity) to 2 and the upper bound to 3.75. For P2 (inlethot_velocity), set the bounds to 1.5 and 1.8 respectively. Click Update
  - Review the candidate points and other graphs, in a similar way to the previous slides (when we had used the Screening Optimizer rather than MOGA)
  - The accuracy of the Screening method depends on the number of samples used. If a sample point does not exist for the optimal condition, the Screening method will not find it. For this reason the Screening method is only good at narrowing the optimization domain but not for finding the optimal condition. Instead, optimization studies should finish with MOGA or NLPQL. With the current settings, the optimizer will retain a set of 100 sample points while iterating several times, removing the worst samples and adding new ones, until 70% of the samples are in the first Pareto front or 20 iterations have been reached

Final Statements

Summary:

- In this workshop you parameterized boundary conditions in FLUENT and expressions in CFD-Post, and explored the features within the DOE, Response Surface and Optimization cells. It was shown that model selection has a significant impact on results so it is good practice to always check the goodness of fit and the maximum predicted error to ensure that your model selections are appropriate. For optimization, a preliminary optimization study was done using the Screening method to narrow the optimization domain and the optimal condition was found using MOGA