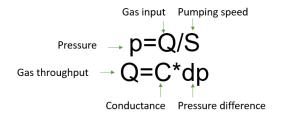
Vacuum simulations (MolFlow+) tutorial for SCC

Marton Ady, Petar Trifunović

12 February 2025

Molflow version: 2.9.27

1) Vacuum equations Two fundamental equations of Ultra High Vacuum:



Units (most used at CERN):

P (pressure): mbar (1 mbar = 100 Pa) Q (gas flow): mbar*l/s (1 mbar*l/s = 0.1 Pa*m3/s) C (conductance): l/s S (pumping speed): l/s

Conductance of a hole of area A:

 $C_{hole}=1/4 * \langle v \rangle * A \qquad \text{where } \langle v \rangle \text{ is the average molecule speed}$ $\langle v \rangle [m/s] = \sqrt{\frac{8RT}{\pi M}} \qquad (M: \text{ gas molar mass in kg, T: gas temperature in K})$

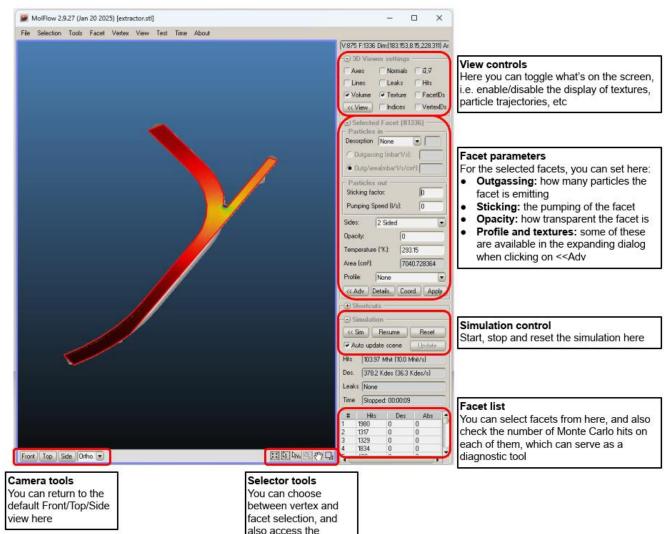
Pumping speed of an area with sticking factor 's':

S = s * C_{hole} = s * ¼ * A * <v>

Local (effective) pumping speed at point B of a pump of speed S_A installed at point A:

 $1/S_{eff,B}=1/S_A + 1/C_{A->B}$ where $C_{A->B}$ is the conductance between the two locations

2) Molflow cheatsheet



Mouse controls:

Selection: left button

Rectangle selection: hold left button

Add/Remove from selection: hold SHIFT to add, CTRL to remove

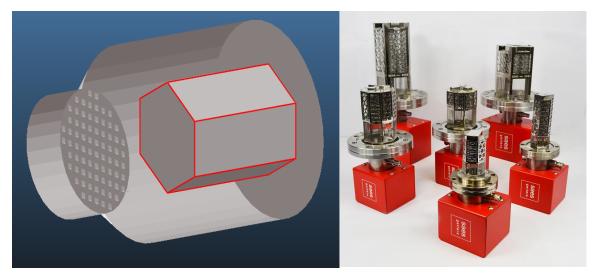
autoscale button

Rotation: right button (slow rotation: hold SHIFT)

Pan: hold middle button (or D on keyboard)

Zoom: mouse wheel (slow / fast zoom: hold SHIFT / CTRL). Alternatively, hold Z on the keyboard and the left mouse button, and drag up/down

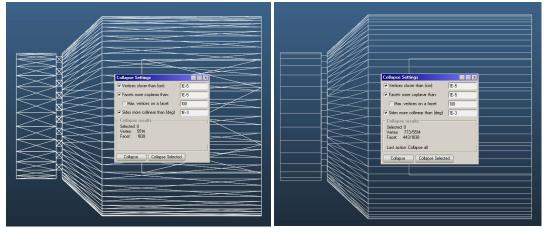
3) Guided work: Effective pumping speed calculation



In this, first part we will approximate the effective (local) pumping speed of a NEG cartridge pump's pumping port. The goal is to use this result in the second part, when we'll be simulating a part of an accelerator.

Opening and collapsing the file

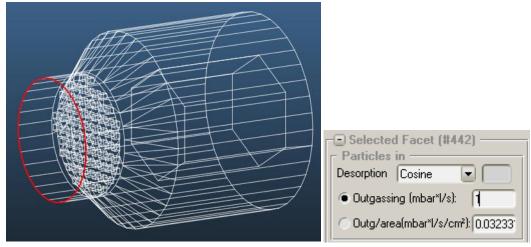
- Load **neg_cartridge_pump.stl**, a file originating from the CAD program, and choose **centimeters** as units. *Hint: STL files are also used by 3D printers, and as such, most CAD programs can export to it.*
- Since this file at its opened form contains only triangles, we need to **collapse** it: leave the default parameters as they are and click collapse:



• You can now save the collapsed file in Molflow format, so during the tutorial you can save it with one click. Choose File/Save as, give the file a name, and press enter – Molflow will append the default .zip extension.

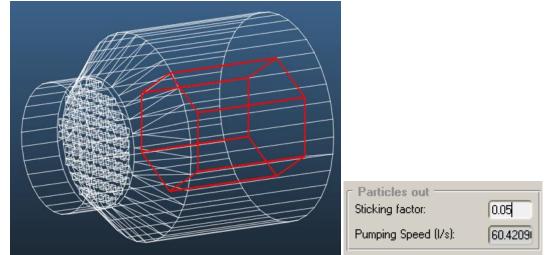
Setting up desorption

• Select the pump entrance facet. On the right side, choose **Cosine** outgassing, then enter **"1"** (equals to 1 mbar.l/s) in the Outgassing field, and click Apply:



Setting up pumping

- In this model, the pumping surfaces are those representing the NEG coating: they are the sides of the hexagonal volume.
- Select the six facets. *Hint:* you can select the hexagonal volume (7 facets) by a rectangle selection then deselect the end cap by a second rectangular selection holding CTRL
- Set sticking to 0.05, a conservative value for NEG. Don't forget to press ENTER or click Apply.



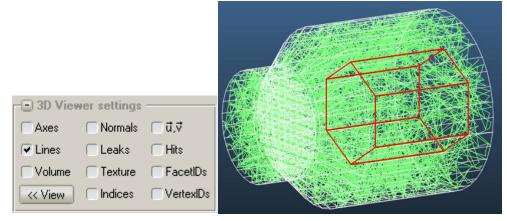
Launching the simulation

• That's it! You have successfully set up outgassing and sticking, therefore all is ready to calculate.



• Click the Begin button:

• Turn on molecule trajectory display (Lines option, upper right) to check if everything is as intended:



Extracting results

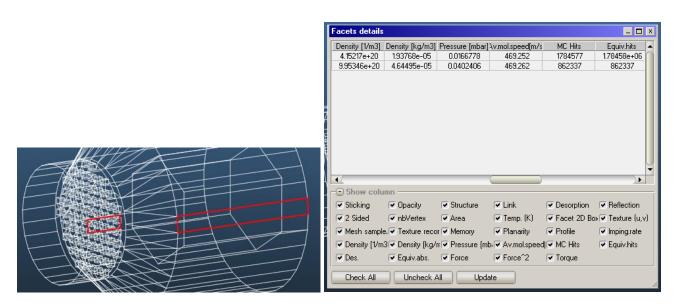
The simulation can be stopped after a few seconds. It's a good time to save results.

We will see four different ways of getting the effective pumping speed:

- By reading the pressure on facets
- By using formulas
- By using profiles
- By using textures

Reading pressure on facets

Select a facet near the entrance and one on the inside of the cartridge volume, the click Details. In the **Facet Details** window, you can read the pressures on the facets:



Staying at the values of the above screenshot, using the first equation of this note:

$$P_{local} = Q / S_{eff}$$

At the entrance: At the pump volume: 0.04 mbar = 1 mbar.l/s / S_{eff} -> S_{eff} = 1/0.04 = **25 l/s** 0.0166778 mbar = 1 mbar.l/s / S_{eff} -> S_{eff} = 1/0.0166778 = **60 l/s**

Small note: you can verify that these values are correct. Select any of the NEG facets. You will see that the pumping speed (corresponding to our sticking factor of 0.05) is 10 l/s. As we have six of these facets, the local pumping speed is indeed 60 l/s.

Using formulas

We can let Molflow evaluate the above equations for convenience. The calculated pumping speeds will be updated every second as the simulation is running, giving us an idea when the calculation has converged.

• In the Tools/Formula Editor menu option, we create new formulas, as in the screenshot:

Formula Editor		
Formula list		
Expression	Name (optional)	Value 🔺
1 QCONST/P436	External pumping speed	24.8505
2 QCONST/P9	Internal pumping speed	59.96
3		
		Ų.
Recalculate now		Move Up Move Down
 Auto-update formulas 		Record convergence
Copy table (all moments)		Open convergence plotter >>
🛨 Format		

Explanation: QCONST is the outgassing, P436 is the pressure on facet #436 (in red)

- We add a second formula. Expression: QCONST/P9 (facet 9 is inside)
- Hint1: you can access a cheat sheet of available formulas by expanding the Format panel

Recalculate now		Move Up	Move Down
Auto-update formulas		🔽 Recor	rd convergence
		Open convergence	e plotter >>
Pn (Pressure [mbar] on facet r	on facet n), Dn (Desorption on facet n), n), DENn (Density [1/m3] on facet n) avg. speed [m/s] on facet n), Tn (temp[

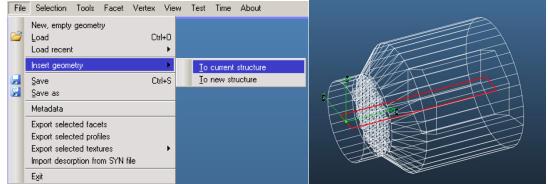
Hint2: if the values in the formulas don't appear or don't update, make sure to check the Auto-update formulas option as shown in the image above

Using profiles

On **rectangular** facets, you can enable profiles which allow quick plotting of the pressure in a direction. Profiles can be set up on existing facets (on the walls), but for our purposes we will use **a transparent facet** that goes through the volume.

Although Molflow has geometry editing options, for simplicity we'll import an extra facet.

• Choose File / Insert geometry / To current structure and open **pump_profile.zip**:



• Set the facet transparent (**opacity: 0**) and **2-sided:**

Sides:	2 Sided 💌
Opacity:	d
- ·	(0)()

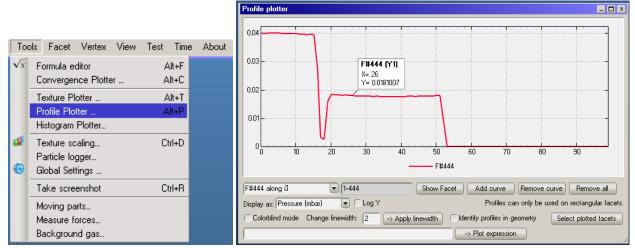
• Enable the **U,V vector** display (upper right), and check that the U vector is oriented along the facet's longer side:

- 3D Vie	wer settings		A	SL/1	A		
C Axes	Normals	₹ ū,⊽				T	
🗍 Lines	🔲 Leaks	🔲 Hits			H		11
C Volume	Texture	🗍 FacetIDs	5-19	COM ST			15
View</th <th>🔲 Indices</th> <th>C VertexIDs</th> <th></th> <th></th> <th>\ge</th> <th></th> <th></th>	🔲 Indices	C VertexIDs			\ge		

• Enable the **profile along the U vector**:

None
None
Pressure/imp/density (ជ)
Pressure/imp/density (∛)
Incident angle
Speed distribution
Orthogonal velocity
Tangential velocity

• Open the profile plotter, chooses the profile you just added and click Add curve, then run the simulation:



- When the simulation is stopped, clicking on the graph reveals the value. You can verify two things:
 - The pressure only drops at the grill (limited conductance)
 - Within volumes, it's almost constant (large conductance)
- Reading the pressure values (0.04 and 0.018 mbar) we can calculate the same pumping speeds

Using textures

Textures are color-coded pressure values displayed directly on the facets.

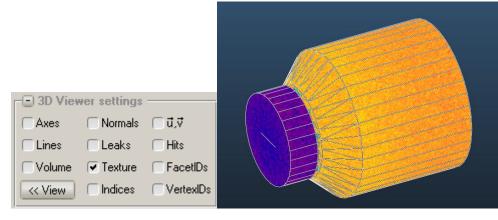
For simplicity, we'll enable textures on all facets.

• Select all facets (by a rectangle or with the shortcut CTRL+A)

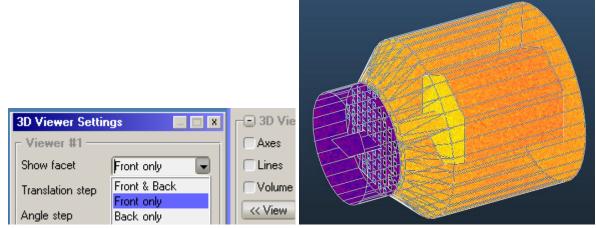
• On the facet parameters menu, open the advanced options by clicking on the << Adv button:

Advanced facet parameters							
Texture properties							
Enable texture Use square cells							
Resolution: 10 cells/cm 0.1 cm/cell							
Number of cells:							
Count desorption							
Count absorption 🔽 Count transparent pass							
Angular coefficient Record direction vectors							
Texture cell / memory							
Memory: 2.31MB Cells: 86510							
Additional parameters							
Reflection: 1 part diffuse, 0 part specular, 0 part cosine^ 0							
Accomodation coefficient: 1							
Teleport to facet: 0							
Structure: 1 Link to: no							
Moving part							
Wall sojourn time							
Attempt freq: 1e+13 Hz Binding E: 100 J/mole							
View settings							
🔽 Draw Texture 🔽 Draw Volume <- Change draw							
Dynamic desorption							
Incident angle distribution							

- Click Enable texture
- Set the resolution to 10 cells/cm, a good compromise between details and speed
- Choose **Count reflection** and **Count transparent pass** as options. The former will count pressure on the walls, the latter on the transparent facet we've added for the profile
- Run the simulation. If you turn on the **Texture** option in the upper right corner, you should see the pressure distribution over the geometry:



• In the advanced viewer options (click the <<**View** button), you can choose to render only the front side of facets, which will allow to peek inside the geometry:



• Finally, to interpret the colors, open the **Texture Scaling** window (**CTRL+D**):

Too	ls Facet Vertex View	Test Time	About	Texture Scaling	
\sqrt{x}	Formula editor Convergence Plotter Texture Plotter	Alt+F Alt+C Alt+T		Texture Range Geometry: Min -inf Image: Autoscale Win: 4.622E-03	1
	Profile Plotter Histogram Plotter	Alt+P		Max inf Moments+const. flow Logarithmic scale Set to geom. Apply min/max. Swap 897KB Max: 4.640E-02	
@ ©	Texture scaling Particle logger Global Settings	Ctrl+D		Gradient	1
	Take screenshot	Ctrl+R		0.005 0.01 0.015 0.02 0.025 0.03 0.035 0.04 0.045	
	Moving parts Measure forces Background gas			Physics mode: Pressure [mbar]	

Here, among others, you can switch on logarithmic view, which is more suitable for plotting large pressure differences.

• Finally, you can extract pressure values from a texture by **selecting a textured facet** and opening the **Texture Plotter** window:

							- 3			
	Text	ure plotter	[Facet #2	4]					- 0	X
	₹\đ	1	2	3	4	5	6	7	8	
	1	0.0173298	0.0180019	0.0184668	0.0168867	0.0193208	0.0182686	0.0173758	0.0184	n
	2	0.0170003	0.0179278	0.0158935	0.0171724	0.0170703	0.0171393	0.01731	0.0176	
	3	0.0171258	0.0167494	0.0168738	0.0175849	0.0178997	0.0176121	0.0159262	0.0178	
	4	0.0181939	0.0174138	0.0166993	0.0168659	0.0179011	0.017239	0.0177211	0.0172	
1	5	0.017224	0.0185446	0.0187321	0.0175725	0.0174115	0.0177305	0.0179663	0.0175	
	6	0.0195167	0.0186393	0.018372	0.0164699	0.0187235	0.0181875	0.0171793	0.0170	
	7	0.0176096	0.0182158	0.017843	0.0173913	0.0179744	0.0179752	0.0168784	0.0166	
	8	0.0171359	0.0169686	0.0164604	0.0180821	0.017903	0.01811	0.0179012	0.0164	
	9	0.0170181	0.0159016	0.0195724	0.0177794	0.0164608	0.0175167	0.0192084	0.0172	
										¥
	40									
										1
		Save	Find Max.				View: Press	ure [mbar]		J
	A	utosize	✓ Autosize	on every up	odate (disabl	e for smooth	n scrolling)			
	_									6

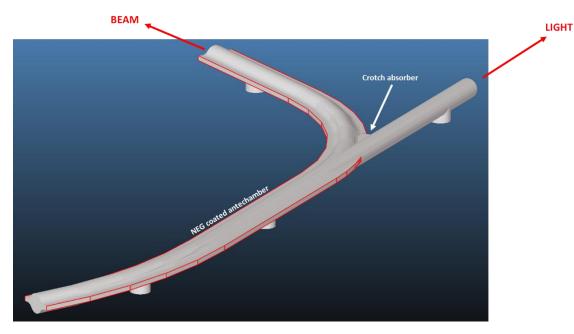
• As seen above, clicking on a cell (or a range of cells) highlights the position on the texture.

It's a good idea to save the simulation now.

You can find the solution in the file cartridge_pump_solved.zip

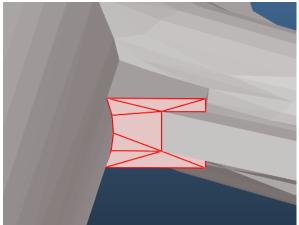
4) Individual work: Accelerator part simulation

The goal here is to use what you've learned in the previous part and apply it to a fictional beampipe with a light extraction port:



Gas input

In a real-life scenario, most gas load will come from the crotch absorber part where synchrotron radiation photons hit the wall.



In our model this is represented by 8 facets, and on these facets the gas load is 0.001 mbar.l/s/cm²

Gas extraction

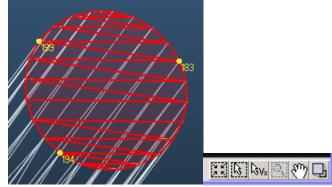
The system has two types of pumping: four times the NEG cartridge pumps whose speed we've calculated in the previous step (25 l/s each), and the NEG coated antechambers, marked above, with a 5% sticking factor.

The goal here is to set up the outgassing, the pumps, then to calculate the pressure in the mid-plane of the accelerator part on your own.

Advice

- You can find the geometry in STL format in the file: extractor.stl where the units are in cm
- You will see that the Collapse command will leave some facets uncollapsed. This is because CAD programs truncate the coordinates of the composing triangles and thus they don't all lie in the same plane. This is a common problem when importing files, and to solve it you can use the Project command to force all facets to be on the same plane. Use this command with the "defined by 3 selected vertex" option, and select 3 vertices on the circle with the Vertex selector tool (lower right corner, see Molflow cheatsheet). Once the

facets are projected the Collapse command should merge them.

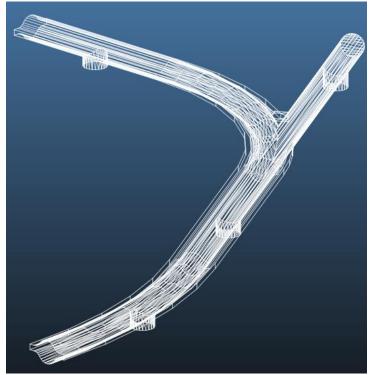


- You don't need to construct the midplane yourself: you can import it from the file **accelerator_midplane.zip** Use the **File / Insert Geometry / To current structure** command. After inserting, you'll still have to make sure it's transparent, and add a texture on it.
- The NEG antechamber is hard to select. Use the Selection / Smart Select option with 12 degrees as option this should allow to select the curved antechamber parts with one click.
- Molflow can memorize selections (groups of facets) to recall them later. When you have selected the antechamber parts, for example, you use the Selection / Memorize selection to / New... command to save the selection (or press CTRL+W). Then you can recall these selections by pressing ALT+1, ALT+2, etc.

4b) Individual work: a possible solution

Importing the geometry

• Open the file extractor.stl with centimeters as units then collapse it:



• We'll see that the end of the light extraction didn't collapse. This is a common problem of using STL files, and its cause is that the CAD program exports coordinates with finite precision, and merging these triangles would create a non-planar facet.



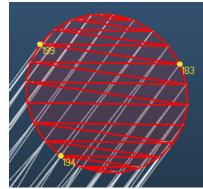
• We can fix the problem by **projecting** all triangles onto the facet's plane. Select the triangles and open the Mirror / Project command:

				Mirror/project selected facets	
				Plane definiton mode	
				⊂ XY plane	
				⊖ YZ plane	
	Fa	cet Vertex View Test Time Abo	ut	⊖ XZ plane	
ł		5.1.		○ Plane of facet # 0 <-Get selected	
		Delete	Ctrl+Del	C Define by 3 selected vertex	
	4-2		Ctrl+N	O Define by plane equation:	
		Shift indices	Ctrl+H		
	4	Facet coordinates		0 *X +0 *Y +0 *Z +0	• 0 // // // // // // //
		Facet details		Mirror facet Copy mirror facet	
		Move			
		Scale		Project facet Copy project facet Undo projection	
		Mirror / Project			
		Rotate			
		Alian to			

- Choose the **define by 3 selected vertex** option
- Switch to Vertex selection mode at the bottom right (cursor with Vx button):



• Select 3 vertices on the end cap (hold SHIFT to add to selection):



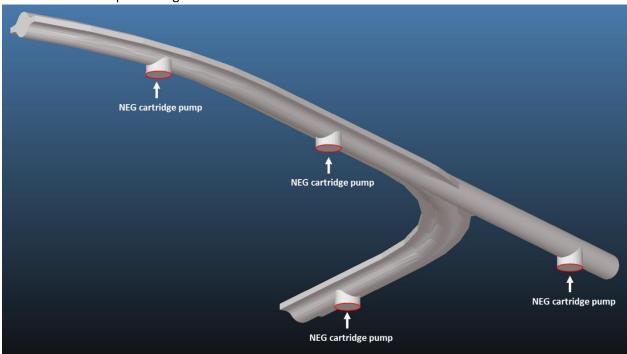
- Click **Project facet** in the mirror / project dialog.
- Visibly nothing happens, however the selected facets have been aligned to the plane. Issue the **Collapse** command again, and the facets will be merged:

Facet	: Vertex Vier	/ Tes	Time	About	
本 (1) 本(1) 本(1) (1) (1) (1) (1) (1) (1) (1)	Delete Swap normal Shift indices Facet coordinate Facet details Move Scale Notate Altign to Split Split			Ctrl+Del Ctrl+N Ctrl+H	
(- - - 	Create shape Create two facel Transition betwe Build intersectior Collapse Explode	en 2		×	

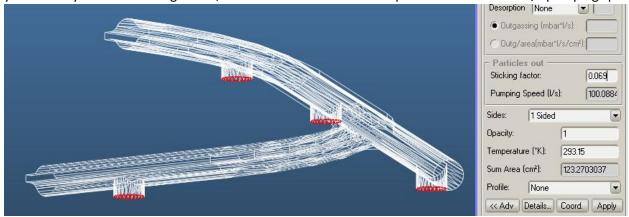
Setting up pumps

First, we will assign the calculated (effective) pumping speeds to the NEG pumps.

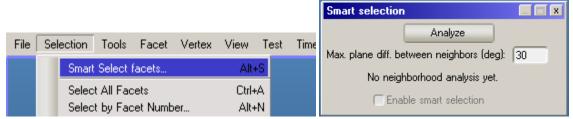
• Select the facets representing them:



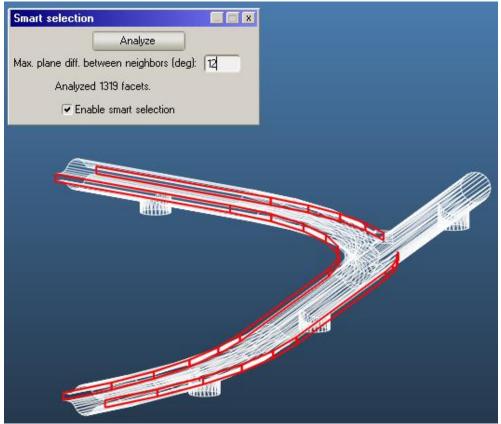
• On the right side, enter the previously calculated 25 l/s pumping speed (if you select the 4 pumps together, you can only edit the sticking factor, enter 0.069 which will correspond to a total of 100 l/s pumping speed):



• Next we'll set up the NEG pumps in the antechamber. For this, we'll use the **Smart selection** dialog (first option in the Seleciton menu):



- The first time we use this option, we need to click **Analyze**, at which point Molflow builds a map to know which facets are neighbors of which other.
- The **degree threshold** defines the max. planar deviation which is still selected for the next facet. For our case, set it to 11 or 12 degrees (found by trial and error)
- Then we can click on one of the antechamber facets and the rest should be selected automatically (hold SHIFT to add to the selection):



• We can save this selection for later: Choose Selection / Memorize selection / New... command (shortcut: CTRL+W) and give it a name, for example Antechamber:

Sele	ection Tools Facet Vertex	View	Test	Time	About			
	Smart Select facets		Alt+S					
	Select All Facets	(Ctrl+A					
	Select by Facet Number		Alt+N					
	Select Sticking							
	Select Transparent							
	Select 2 sided							
	Select Texture							
	Select by Texture type							
	Select Profile							
	Select Abs > 0							
	Select Hit > 0							
	Select large with no hits							
	Select by facet result							
	Select link facets							
	Select teleport facets							
	Select non planar facets							
	Select non simple facets							
	Invert selection		Ctrl+l					
1	Save / Overwrite selection							
11	Select memorized			A	dd new	L	∑trl+W	Catalog Manager and
	Clear memorized							Enter selection name
								Selection name Antechamber
	Select Description							
	Select Dyn. Desorption (Outg.M	ap)						OK Cancel
	Select Reflective							

• Now you can recall this selection group anytime by **Selection / Select memorized / Antechamber** (shortcut: **ALT+1** for the first selection group, **ALT+2** for the second, etc.):

Sel	lection	Tools	Facet	Vertex	View	Test	Time	About	
	Smart	Select I	facets		Alt+S				
Ð	Selec Selec Selec Selec Selec	t Stickin t Transp t 2 sideo t Texture	set Numb g arent d		Ctrl+A Alt+N				
		t Profile	ture type						
	Seleci Seleci	-	-						
	Select Select Select	t non pl	t facets anar face mple face			Ctrl+I			
	Save	/ Overw	vrite seleo	ction		۰,			
		t memor memoriz))		ntechamber elect previous	Alt+1 Alt+F11
	Select	t Desorp t Dyn. D t Reflect	esorption	ı (Outg.Ma	ар)			elect previous elect next	Alt+F11 Alt+F12

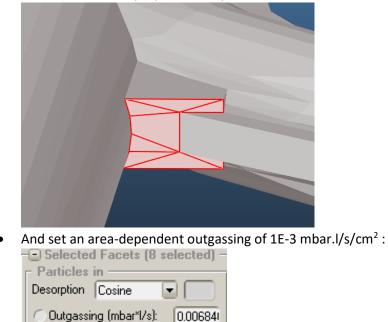
• Finally, set a sticking factor of 0.05 for these facets:



Setting up the desorption

For simplicity, this model will have only one gas source: the crotch absorber.

• For this, select the perpendicular part of the crotch absorber:

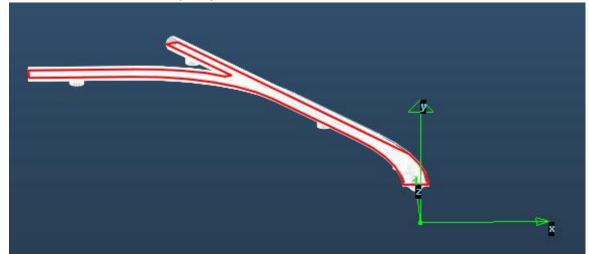


Adding texture

You can import the half-plane of the accelerator from an external file:

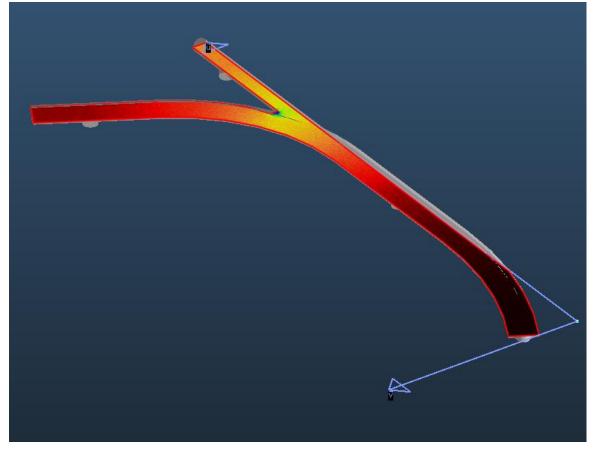
- Choose File / Insert Geometry / To current structure and insert accelerator_midplane.zip
- Set it to 2-sided and set its opacity to 0

Outg/area(mbar*l/s/cm²): 1E-3



• Add a texture with Count transparent pass option. Recommended resolution: 3 cells/cm

• Turn on texture display (upper right) and choose **Front only** view option in the advanced view settings:



You can adjust the texture scaling in the dialog (opens with CTRL+D).

At the end of the job, you should get a texture similar to the above.

Molflow website: https://molflow.docs.cern.ch/

Tutors' contact: marton.ady@cern.ch, petar.trifunovic@cern.ch