TEAM LOG BOOK

ME421 Final Project: CFD Analysis

Mode Frontier

Justin Dilworth, Cameron Hjeltness, Dylan Rinker

Entry by: Dylan Rinker

For help with the CATIA node:

- Help
 - Integrations
 - CATIA Node

The PATH variable needs to be able to find the CNEXT.exe

Have CATIA Automatically update part and automatically update product checked:

- Tools
 - Options
 - General
 - Parameters and Measurements
 - Measure Tools Tab

Entry by: Justin Dilworth

Setting up Parameters in Catia

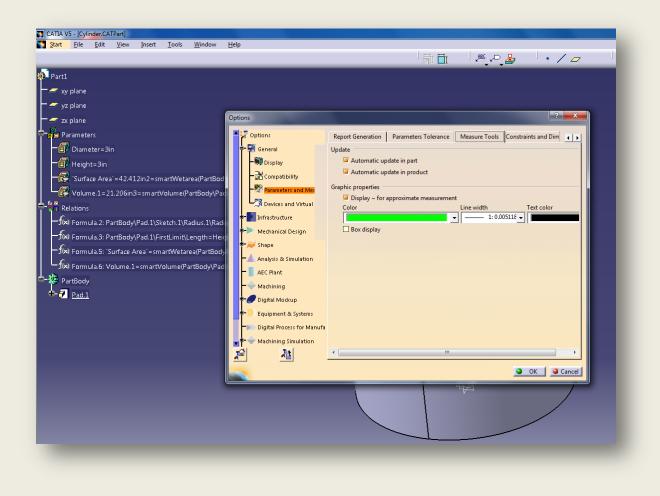
- To make Parameters, you first open up Catia Formula.
- After entering the formula setup, There is a button in the lower left Labeled <u>New Parameter of</u> type
- Before you hit the button first select the type of parameter you want, such as length, area, volume, etc.
- After you make your parameter, it will show up in the space above as length.1, length.2, or volume.1, etc.
- You can change the name from length.1 to whatever you want
- Now you need to decide if you want the perimeter to be driving or driven.
- For making the perimeter a Driving the Feature (length most common used)
 - First select the feature you want to drive, such as Radius or length of the part in the list of parameters above
 - \circ $\;$ Then click the add new function button
 - There Once there find your parameter you created and make sure that it is in the equals to segment on top
 - $\circ \quad \text{Then click ok} \\$
- For make the perimeter a Driven Feature (surface area, volume, stress etc.)
 - o First select your parameter you just created
 - Then click the add function button
 - o Then set your parameter equal to what you want displayed and click ok
 - You will get a popup window and click ok

• Your driven parameter should have a graduation hat looking thing in front of it.

Other Presets

Options /Parameters and Measures /Measure Tools

Check: Automatic Update Part, Automatic Update Product



Entry by: Cameron Hjeltness

Today, we got a lot of work done as a group. Prior to this, we've been doing independent research. This shows that if we need to get more work done, we need to work in a whole group every time. This way, we can bounce our ideas off of each other and learn from each other's discoveries.

Accomplishments for the day include:

Setting up tentative schedules, both general and specific.

Workflow Nodes

ariable Nodes

CAD Nodes

Goal Nodes

🗞 🏷 💸

🏘 📃 🗊 🎐

🖉 🐼 🧏 🗶 💸

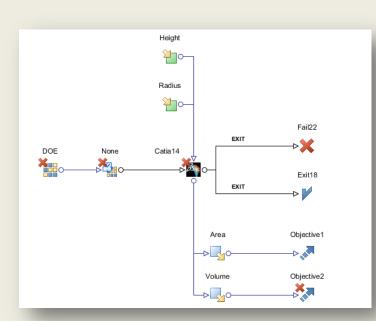
1.11

331 🕵

- Defined team goals and deliverables
- Began documenting work. Lost documentation data is negligible.
- Learned how to effectively define parameters in CATIA. (See entry by Justin Dilworth)
- Learned how to set up the CATIA node. (See entry by Dylan Rinker)
- Learned the basic set-up requirements for Mode Frontier. (See below for details)
- Found online tutorials for Mode Frontier 4.
 - wn.com/modefrontier
 - Youtube: Username: DanilNagy

Basic set-up for logic nodes:

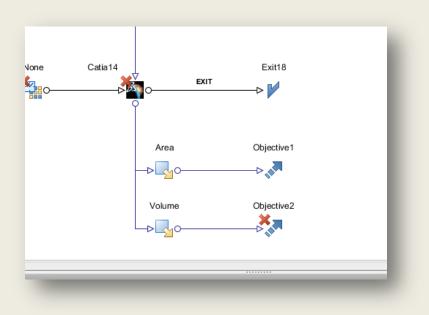
- A basic set up is shown in the diagram below.
 - All set-ups will have the following components:
 - A scheduling node
 - Logic End nodes
 - Does not necessarilly need both
 - Variable nodes such as Input and outputs
 - A program node. Shown below is a CAD node for CATIA
 - A goal node. Shown are Design Objective nodes.



The purpose of to take inputs

this set-up is such as

"height" and "radius" and supply them to Mode Frontier. The CAD node then signals M.F. to then open the specified program, applies the parameters, and then optimizes the solid model based on the objective nodes.



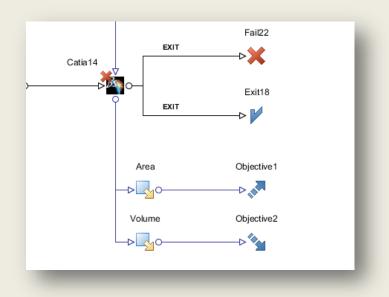
Shown above are the objective nodes. An arrow up signifies that you are maximizing a value. Typically you will have conflicting objectives, allowing M.F. to solve for a solution. Note that one objective is without a red x, and the other has a red x present. This means that Objective 2 has not yet had its properties specified.

Objective Properties - 4.2.1 b20100318		×
Objective Properties		
Name	Objective2	
Description		
Enabled		
Format	0.0000E0	
Objective Expression Properties		
User Expression		
Туре	Minimize	•
	Maximize	
	Minimize	
Data Input Connector		
🖳 Volume		
OK	Cancel	Help

Double click the objective to enter objective properties. You can edit the objective statement with the drop-down menu and select either maximize or minimize.

Variables	Expression	ı				
/olume	1 Volume					
	Basic Fund	tions			Operator	'S
	sin	COS	tan	degToRad	~ &	
	asin	acos	atan	radToDeg		
	log	In	exp	sqrt	<u>%</u> ,	
	abs	sgn	rand	pow	7 8	
	ceil	floor	round	mod	4 5	
	min	max	interp	vect		

Next, to define the user expression, click the calculator icon above the previously mentioned drop-down menu. Here, you will enter the expression editor. On the left is a list of your variables that the current node deals with. Select the variable you wish to control the node. You can also introduce some more algebra to the node. For now, we will leave the variable as is. Click "Apply"



Notice the difference now for the objective node. The red x has been removed and the node is now defined. The arrow is also pointing down denoting that the objective here is to minimize the volume of the object in question.

4/8/11

See page 16 for the solution to this error!

Entry by: Cameron Hjeltness

	Performing Introspection	California -			
Description Document	📹 🐐 🚔 🖡 🗟 💲 Auto Scroll On			tudent Submissions	Monday 📾
Script File					
s relative	<u>کم</u>	100%			
CATIA Document Advanced Pro	Project/Test	t Parts\Part1.CATPart			
CATIA Work Space Properties					
Windows Only Properties	Apr 08 2011 14:59:10:984 Introspectio				
Export	Apr 08 2011 14:59:10:984 Introspectio	in script: el0743\.modeFRONTIER\4\tmp\ft	mn 2011 04 08 1456	15 42	
Screen capture			mp_2011.04.00_14.00.4	··	
ouroun capture	Apr 08 2011 14:59:10:984 Introspectio	n result lile: el0743\.modeFRONTIER\4\tmp\ft	mp 2011 04 08 14 56	45 47=	
	Apr 08 2011 14:59:10:984 Begin intros			····	
	-				
	Apr 08 2011 14:59:11:015 java.io.IOExe "C1Usershi	ception: Cannot run program "cni jel0743\.modeFRONTIER\4\tmp\1		45.4	
Process Input Connector		ess error=2. The system cannot		+3_4	
Scheduler					
Data Input Connector					-
Height	Abort		Close		
Radius		ee war			<u> </u>
		□ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □			66
ОК		Cancel		Help	
				neip]
UK .					
UK					
	_				
UK	_				
UK	~				
UK	~				

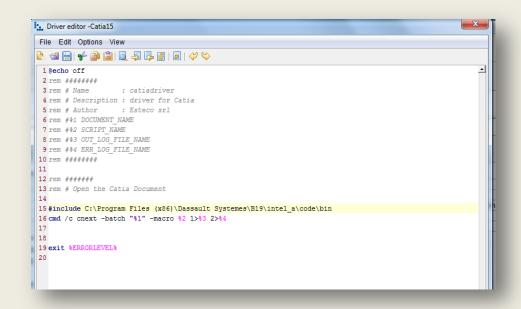
Worked on setting parameters in M.F.

We encountered this problem yesterday but couldn't fix it. For some reason, M.F. can't run the program inside of our personal drives. I believe that this is due to the fact that we aren't administrators. We only have administrator access to start the file with the modified batch program, but once we are in the program itself, these rights are revoked.

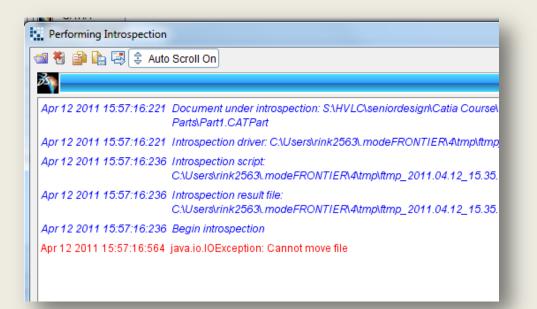
Will probably have to talk with Cam or Alec to get this fixed. Maybe we need to be given administrator rights? Maybe we are missing something.

4/12/11

Group Entry



Tried re-routing the path directory with some C++ code...



It didn't work, so we tried a different computing language.

F	le Edit Options View	
P	⊴ ⊨ + ≱ 🖹 🧕 🖧 🗄 🕘 🖉 🛠	
	@echo off	-
	2 rem ########	
:	rem # Name : catiadriver	
	rem # Description : driver for Catia	
	rem # Author : Esteco srl	
	rem #\$1 DOCUMENT_NAME	
:	rem #\$2 SCRIPT_NAME	
	tem #\$3 OUT_LOG_FILE_NAME	
	erem #\$4 ERR_LOG_FILE_NAME	
1) rem ########	
1:		
1:	2 rem #######	
1:) rem # Open the Catia Document	
1	i de la constancia de la c	
1	<pre>idjsp:directive.include file="C:\Program Files (x86)\Dassault Systemes\B19\intel_a\code\bin" /></pre>	
1	cmd /c cnext -batch "%1" -macro %2 1>%3 2>%4	
1		
1		
	exit %ERRORLEVEL%	
21		

Here we tried rerouting the path with Java script. It didn't work either.

frontier.bat	
_	
	Set GUI Actions
	Load GUI Resources
modeFRUNIIER:	Load Scheduler Plug-ins
modeFRONTIFR:	Load RSM Plug-ins (00 109 sec)
mode FRONT I ER:	Load RSM-U Plug-ins
modeFRONTIER:	Load MCDM Plug-ins
modeFRONTIER:	Load SOM Plug-ins
modeFRONTIER:	Load Hierarchical Clustering Plug-ins (00.047 sec)
modeFRONTIER	Load Partitive Clustering Plug-ins (00.078 sec)
modeFRUNIIER:	Load MDS Plug-ins
	Init Workflow Fanel
mode FRONTIER:	Init Bun Analysis Panel
	Layout GUI Components
modeFRONTIER:	GUI Start Up Time
modeFRONTIER:	Total Start Up Time
2563\.modeFRON \introspection ecified	pption: Cannot run program "cnext.exe" (in directory "C:\Users\rink ITIER\4\tmp\ftmp_2011.04.12_15.35.58_418\workflow\nodes\Catia15\tmp 44183"): CreateProcess error=2, The system cannot find the file sp
at jav	esteco.util.system.NativeProcess. <init>(NativeProcess.java:61)</init>
at it.	esteco.integration.catia.workflow.gui.swing.CatiaParameterChooser.
	meterChooser.java:500)
	esteco.integration.catia.workflow.gui.swing.CatiaParameterChooser.
	on(CatiaParameterChooser.java:651)
	esteco.integration.catia.workflow.gui.swing.CatiaParameterChooser.
	tiaParameterChooser.java:87) esteco.integration.catia.workflow.gui.swing.CatiaParameterChooser\$
IntrospectionT	'hread.refreshIntrospection(CatiaParameterChooser.java:927)
THOI COPCCCIONI	

We looked at the batch file, the program given to us in order to even begin running M.F., and ultimately we decided that we don't have sufficient user privileges. We need to contact Alex Odom for further assistance. Until we can take care of this issue, we seem to be dead in the water.

4/14/11

Entry by: Cameron Hjeltness

Worked on research for the Star CCM software.

We're a little behind schedule... finally have two functioning computers to work with that don't seem to be giving us problems so we should be able to get some good work done!

Accomplishments for the day:

- Research research research...
- Found a good tutorial site: www.computationalfluids.com/
- Has 5 good tutorials on:
 - o Flow around 2D cylinders
 - Flow around a bullet
 - Flow over a wing
 - A ship in a channel
 - Flow through a supersonic nozzle... sounds cool
 - Also has an awesome section on CFD theory
 - o Textbook in PDF format
 - o Text on applied computational aerodynamics
 - o CFD in Excel
 - CFD using probability methods (not going to use this too much for the project I would think)
 - Something called "Monte Carlo Methods"
- As if this was enough the site also has publications for practical applications for CFD analyses:
 - o Boiler design
 - o Pipe Erosion
 - o Screw compressor Design
 - Vehicle Aerodynamics
- Learned how to set the mesh with some of the given parts free to download from the website, the controls of the software, different types of boundary layers, boundary constraints.

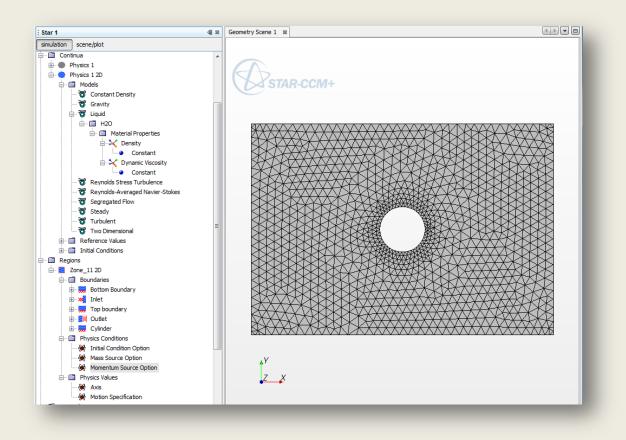
4/16/11

Entry by: Cameron Hjeltness

Found another video tutorial on Flow field simulation, tracking particles in flow, and calculating efficiency:

http://www.youtube.com/watch?v=DHJ2Pq8D9Uo

Worked on the first tutorial from the computationalfluids.com website, but got stuck on setting boundary conditions for the fluid. The tutorial shows something in the design tree that I can't find at all... Here is as far as I got:



Progress for the tutorial:

- Converted the 3D geometry to 2D to save calculation time
 - o Tutorial required it but only because they had the limited trial software
- Set the mesh
- Set boundary conditions (Velocity inlet, pressure outlet, and rigid boundaries)
- Set fluid properties
- Set physics properties (gravity, constant density/viscosity for the liquid, etc...)

I moved on to the video tutorial to try making progress.

	: Output -	Star 1																	
		-		ver: starc HKEY_LOCA		-	E\MPICH\SN	MPD\process	\2404	registry	key,	error	5,	Access	is	denied.			
	Unable	to oper	the	HKEY_LOCA	L_MACHINE	C\SOFTWAR	E\MPICH\SN	MPD\process	\3268	registry	key,	error	5,	Access	is	denied.			
	Unable	to oper	the	HKEY_LOCA	L_MACHINE	2\SOFTWAR	E\MPICH\SN	MPD\process	\4448	registry	key,	error	5,	Access	is	denied.			
	Unable	to oper	the	HKEY_LOCA	L_MACHINE	C\SOFTWAR	E\MPICH\SM	MPD\process	\4852	registry	key,	error	5,	Access	is	denied.			
	Unable	to oper	the	HKEY_LOCA	L_MACHINE	C\SOFTWAR	E\MPICH\SN	MPD\process	\4956	registry	key,	error	5,	Access	is	denied.			
	Unable	to oper	h the	HKEY_LOCA	L_MACHINE	C\SOFTWAR	E\MPICH\SN	MPD\process	\4316	registry	key,	error	5,	Access	is	denied.			
	Unable	to oper	h the	HKEY_LOCA	L_MACHINE	C\SOFTWAR	E\MPICH\SN	MPD\process	\3456	registry	key,	error	5,	Access	is	denied.			
*	Unable	to oper	the	HKEY_LOCA	L_MACHINE	C\SOFTWAR	E\MPICH\SN	MPD\process	\4696	registry	key,	error	5	Access	is	denied.			
\square	MPI Dis	tributi	on :	MPICH2:ss	m														
	Process	rank () ME-(GJ115-S02.	ad.uidaho	.edu 4352	2												
				GJ115-S02.															
				GJ115-S02.															
				GJ115-S02. GJ115-S02.															
				GJ115-S02. GJ115-S02.															
?				GJ115-S02.															
	Process	rank 7	ME-0	GJ115-S02.	ad.uidaho	.edu 4308	В												
		_	-						_			_	_				_	_	

I am hoping that these error messages won't prevent my work later on in this tutorial. Its very frustrating trying to learn a prgram and fighting it all the way... Why doe we have the software if we can't use it? Its very counter-productive. I plan on talking to Dr. Odom or Dr. Beyerlein to potentially get administrator access, although I know that this issue originates in the I.T. department...

4/17/11

Entry by: Cameron Hjeltness

I tried importing the previously created geometry that we made in CATIA, our simple cylinder, but it gave me the following error messages:

Star 1 - STAR-CCM+		۲ĭ
File Edit Mesh Solution Tools Window Help		
7 = II • • E 🛱 🖓 🖓 II • II = 7	🕻 🗆 🔻 🏘 🗑 🕅 🏧 🔯 📖 🔯 🛄 🚳 🦘 🥐 🧶 🏂 🚥 🜒 👯 🖾 🗔 🐘 🕨 👘	
: Star 1 🕘	3D-CAD View 1 🗱	
simulation scene/plot 3D-CAD 3D-CAD Model 1 Bodies Bodies Image: Features Features Bodies	CISDR CCM+	
ー 贞 xY 一 贞 YZ - 人 7X	irror	
EnportCad 1 Design Parameters	Exception Could not find appropriate native CAD importer license.	
		₩ ₩
Close 3D-CAD Update 3D-CAD	<pre>(Address: 000000140004CA2) , ERROR: SymGetLineFromAddr64, GetLastError: 487 (Address: 000000140004CA2) , 000000140004CA2 (star-ccm+): (filename not available): (function-name not available) , ERROR: SymGetLineFromAddr64, GetLastError: 487 (Address: 00000007765652D) , 00000007765652D (kernel32): (filename not available): BaseThreadInitThunk</pre>	
Properties	, ERROR: SymGetLineFromAddr64, GetLastError: 487 (Address: 00000000779EC521)	
Grid spacing 0.01 m	, 00000000779EC521 (ntdll): (filename not available): RtlUserThreadStart	
3D-CAD Model 1	error: Server Error	-
	<	•

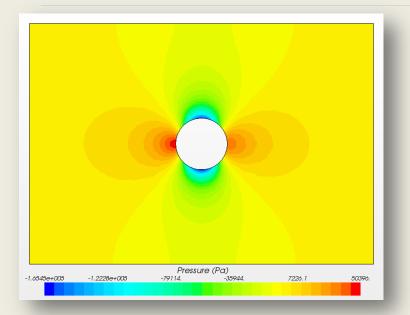
I don't think this will be too much of a problem since I can create simple geometries in Star CCM but it would be easier to just import our geometries that we already have made.

NOTE: This will be a big problem later on if we can't import other complex geometries like the F1 muffler, or any other part like that.

Today I plan on finishing the tutorial from last night

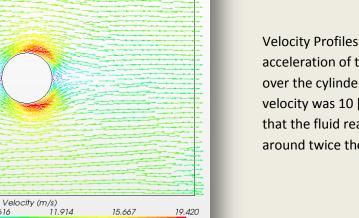
I talked with Dr. Beyerlein about the issue that listed above and he told me that I should switch computers. I don't know if it was this that enabled me to get the tutorial working or not because I also started again from scratch but I finally got things working the way they were suppsed to!!

Here are the results of a simple CFD analysis for a 2D cylinder in cross flow:



A graph of Absolute pressure.

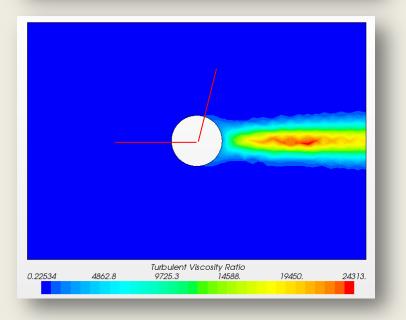
NOTE: Pressure is high at the very leading edge of the cylinder and extremely low at the tops. This is very logical as the water hits the cylinder very hard at the front and has to speed up to curve over the top.



19.420

15.667

Velocity Profiles giving a sense for the acceleration of the fluid as it passes over the cylinder wall. The input velocity was 10 [m/s] and this shows that the fluid reaches a peak vlocity of around twice the input.



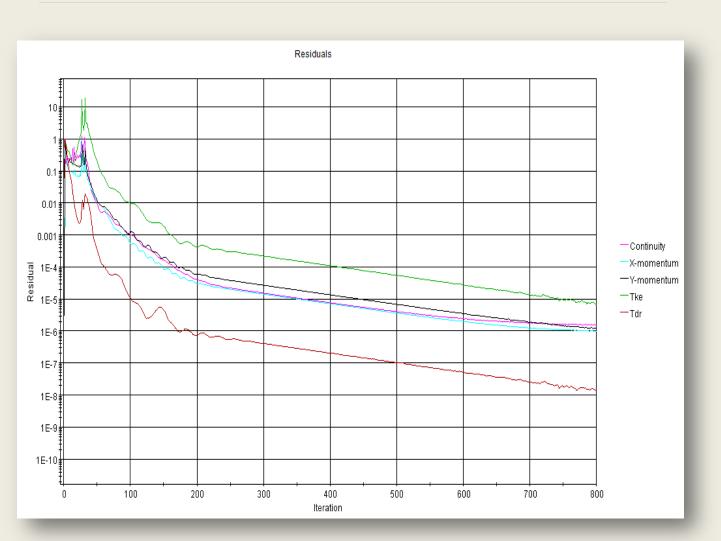
0.65616

4.4089

8.1616

A chart for turbulent viscosity ratio. I believe this is a ratio of the viscosity for the turbulent region versus that of the undisturbed fluid region

Interesting to note, this shows the relative separation point for the flow of the cylinder. Its at a point of about 100 degrees from the negative x-axis.



A chart of the residuals.

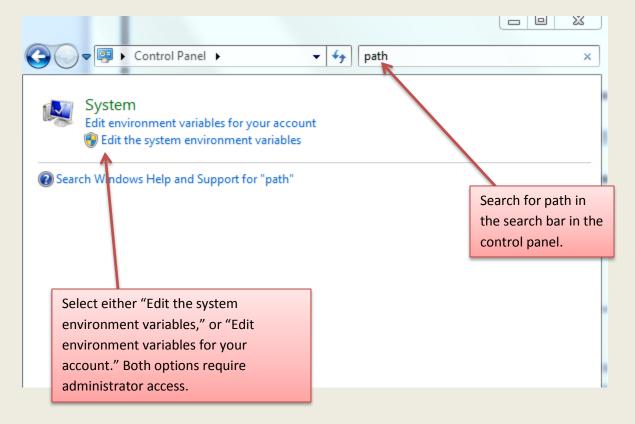
Displays:

- Continuity
- Momentum in the x-direction
- Momentum in the y-direction
- Tke (Turbulent kinetic energy)
- Tdr (Turbulent dissipation rate)

4/18/11

Entry by: Dylan Rinker

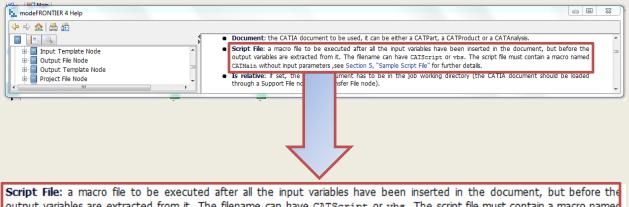
Refering to the original error we had concerning the CNEXT.exe application, we found that the Windows Environmental PATH variable needs to have the file path to the location of the CNEXT.exe program. The Environmental variables are in the control panel. You have to search for them in the search bar.



The Environment Variable Diologue box will pop up. Scroll down in the System variables section to find the "Path" variable. You will then select it and add a ; after the existing text, then add the file path to the location of CNEXT.exe.

nvironment Varia	ables S							
User variables f	for rink2563							
Variable Value								
TEMP	%USERPROFILE%\AppData\Local\Temp							
TMP	%USERPROFILE%\AppData\Local\Temp							
	New Edit Delete							
System variable	s							
Variable	Value							
OS	Windows_NT							
Path	c:\Program Files (x86)\WVIDIA Corporat							
PATHEXT	.COM;.EXE;.BAT;.CMD;.VBS;.VBE;.JS;							
PROCESSOR_	A AMD64 🔻							
	New Edit Delete							
	OK Cancel							
Scroll dov	wn to the Path variable,							
select it,	click edit, and then add							
	ath to CNEXT.exe after the							
	ext. Remember to add a							
-	n (;) before your new file							
path.	in (,) before your new me							

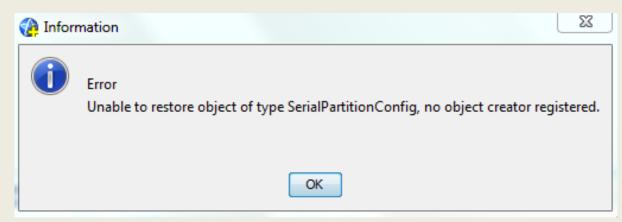
After successfully accessing the parameters of our CATIA part through the CATIA node in modeFRONTIER, we have ran into another stumbling block. It appears that a CATIA script file is required to link the input and output parameters with the constaints set by modeFRONTIER. The help menue is fairly concise about this, but the problem is that none of our group knows even what a CATIA script is!



output variables are extracted from it. The filename can have CATScript or vbs. The script file must contain a macro named CATMain without input parameters ,see Section 5, "Sample Script File" for further details.

Section 5 with the sample script gives some good examples and would be helpful, but none of us know in what context to use it, or even how to approach this issue. We have contacted Alex Odom again (he successfully helped us figure out the Path variable problem), and hopefully we will be able to actually preform a simple optimization of our CATIA part in modeFRONTIER.

I attempted to use the Transonic flow over a bullet Star CCM Tutorial, but ran into some issues. First a diologue box popped up saying it was unable to restore some object that had to do with the Serial Partition Configuration. I am not sure what this means, but I will be sure to ask Alex.



The only other thing that happened when I attempted to access this tutorial was the appearance of some code in the output window.

Output - blunt_body Starting local server: starccm+ -server "S:\\HVLC\\seniordesign\\Catia Course\\10. Spring 2011\\Final Projects\\CFD\\Star CCM Tutorials\\Transonic flow over a b Serial process 3008 STAR-CCM+ Version 6.02.007 (win64/intel11.1) License version: 23 Oct 2009 Required feature version set to 2011.02 or later Checking license file: 1999@cdadapco.engr.uidaho.edu 1 copies of ccmpsuite checked out from 1999@cdadapco.engr.uidaho.edu Feature compsuite expires in 266 days Server::start -host ME-GJ115-S03:47827 Loading simulation database: S:\HVLC\seniordesign\Catia Course\10. Spring 2011\Final Projects\CFD\Star CCM Tutorials\Transonic flow over a bullet\blunt_body.sim Loading module: MaterialModel Loading module: MotionModel Simulation database saved by: STAR-CCM+ Version 1.04.004 (Windows/intel) Tag date = 12/01/2004 3:20 PM Loading into: STAR-CCM+ Version 6.02.007 (win64/intel11.1) Serial Tag date = 2011-01-31 18:56:34 GMT Simulation database load aborted. Unable to restore object of type SerialPartitionConfig, no object creator registered. Command: RestoreState In: [Machine::main, SimulationStateRestorer::restore, SimulationstateRestorer::performRestore, SimulationStateRestorer::reconstructObjects] error: Server Error Closing connection server and exiting The necessary objects do not seem to be accessible for this tutorial,

possibly because of user privileges.

/19/11	See page 21 for solution to importing license issue
ntry by: Group CATIA V5 - [Analysis1]	In order to do 3D CFD analysis in Star-CCM with a file from Catia, an extension of Catia called STAR-CAT5 is required to use a function called mesh pipeline. We do not have this add-
Start <u>F</u> ile <u>E</u> dit <u>V</u> iew Insert <u>T</u> oo	on apparently.
 Sketcher Part Design Assembly Design Drafting Advanced Machining Generative Structural Analysis Infrastructure Mechanical Design Shape 	 It is site has not been analyzed yet Notify us hing STAR-CAT5 Is for launching STAR-CAT5 are as follows: Intervention of the start of the st
Analysis & Simulation	Advanced Meshing Tools Simulation > STAR-CAT5.
AEC Plant Machining Digital Mockup Equipment & Systems Digital Process for Manufacturing Machining Simulation Ergonomics Design & Analysis	Generative Structural Analysis Generative Structural Analysis Core guic grew Insert Iools Analyze Window Help Infrastructure Mechanical Design Shape Analysis & Simulation Knowledgeware Generative Structural Analysis

Questions for Alex tomorrow:

What is a Catia script file, how do we make one, and how do we use it in the CATIA node of modeFRONTIER?

Is there a way to get/use the STAR-CAT5 add-on to be able to do 3D analysis of CATIA parts in Star-CCM?

When we try to import the volume or surface mesh from a CATIA we get this message:



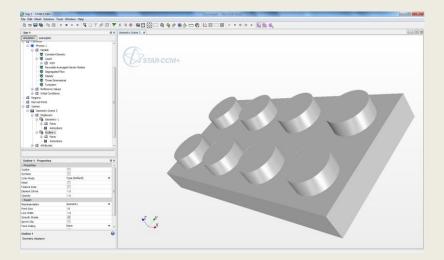


Entry by: Justin Dilwoth

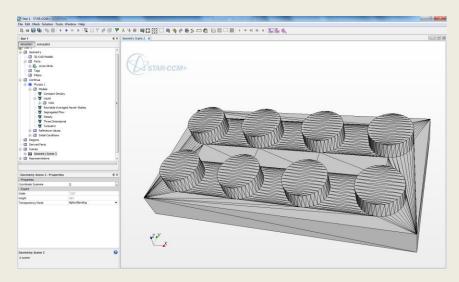
The goal was to import a catia model into Star CCM so I could create an environment to run a 3d simulation of a fluid flowing over a catia part.

My Plan of Attack:

I tried to import a catia part as an .stl file so that Star CCM could be imported as a surface mesh, the create and environment with set parameters. This ended up solving our issue for the importing license. This seems to be a good way to get around this error for future use.



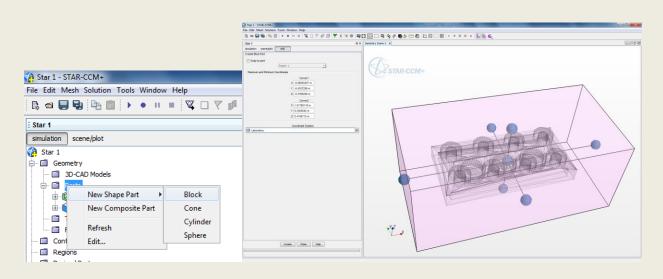
Here is a part that I managed to import as an .stl file into Star CCM. It appears to load the part as a surface within a geometric scene that it automatically created.



Here is the mesh of the Part created by catia when it saved the part as an .stl part. Not Shure if this is a mesh we can or can't use to create a simulation.

Maybe use a .stp file extension to generate a better mesh?

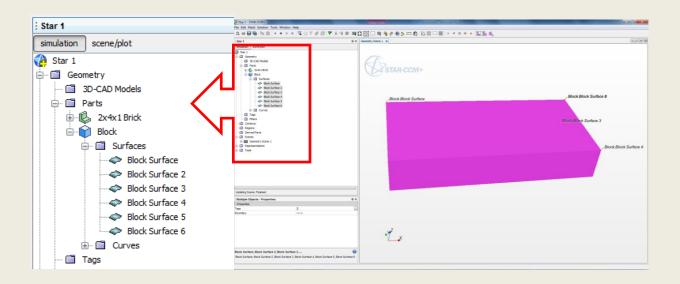
I believe that the next step in the simulation was to create an environment in Star CCM in order to do a 3d flow simulation.



By going to the part folder in the design tree in the left, and right clicking, and selecting new shape then block as shown above. When the part is imported created in Star CMM it creates it as one surface, so in order to break up the surfaces so I can select inlet and outlet parameters with the simulation I go to the next step I discovered.

Star 1 simulation scene/plot Star 1 Geometry 3D-CAD Models Parts Star 1 Surfaces Surfaces Star 1 Surfaces Surfaces Surfaces Star 1 Tags Filters Continua Regions Derived Parts Scenes Geometry Scene 1 Representations Tools	Highlight Combine Create New Part From Surfaces Split Non-Contiguous Split By Part Curves Split By Angle Split By Patch Set Boundary Apply Tag	Split Part Surfaces by Ang Search by Name	문 Tree View	E List View
	Rename		Select all	Clear selection
	Delete	Angle (degrees) 89		
	Edit			Apply Close Help

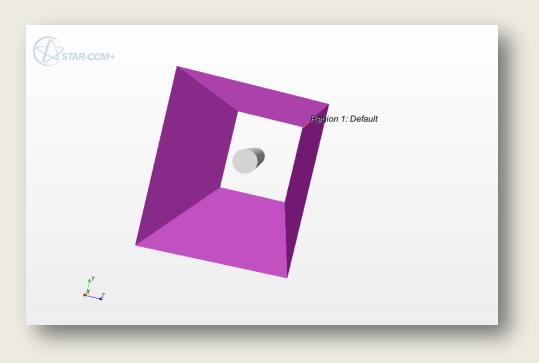
If you go to the design tree and select the block surface and use the split by angle. I believe this feature looks at two surfaces and looks to see if the angle Is greater than 89 degrees, and if so beaks the surface into more surfaces.



The results from breaking the surfaces up are as shown above. I believe that this will potentially allow me to set parameters within Star CCM to create a simulation.

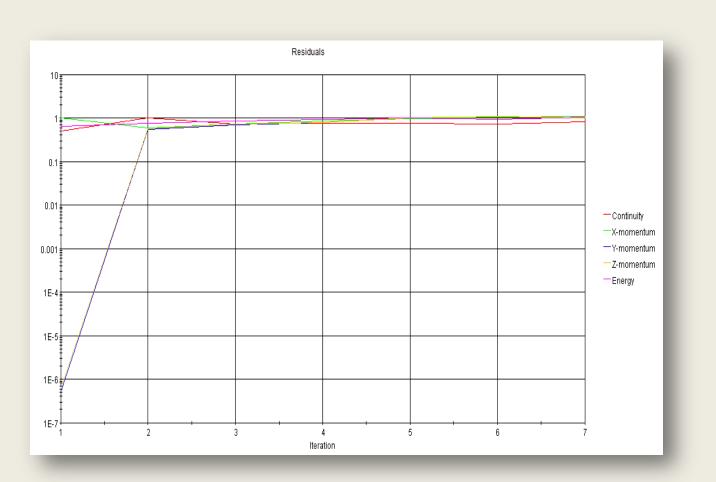
4/20/11

Entry by: Group



Showing how we assigned the various boundary conditions. Here, we selected the highlighted surfaces as walls.

STAR-CCM+			
[<u>z_x</u>	Vorticity: Magni	tude (/s)	

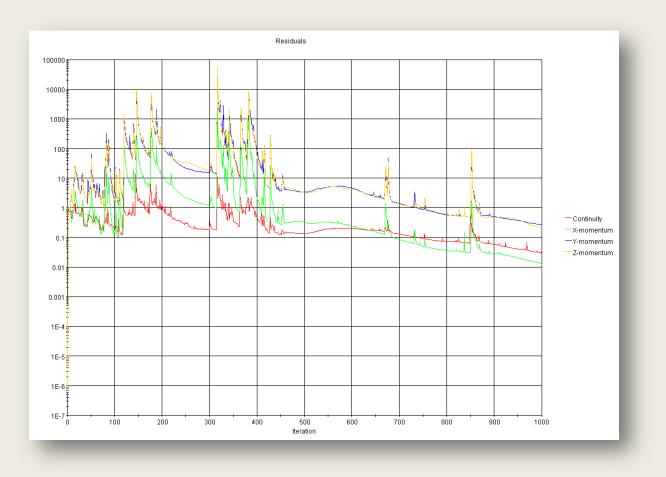


6 iterations for a test... Didn't work but we realized that we hadn't selected the cylinder as a wall within the mesh. So there wasn't anything to solve. Will try again.

A possibility is that we should have let the computer run more iterations. The results seemed to converge nicely but we're not even sure what the residuals are saying, so it's hard to say if the convergence was good or bad.

Second try....

We selected the cylinder inside the boundaries as a wall. This should help our results.



The residuals look quite different, but the program never did reach a solution, even after 1000 iterations.

Talked with Dr. Beyerlein and he suggested using a simpler geometry. He also noted something that I never considered, but that makes a lot of sense. We should cut the edges of the cylinder, otherwise the geometry will have an infinite radius, something that computer programs hate dealing with.

De
Star 4 Star 4 Star 1 Star
Start Converts Book Start <
Sector State Societaria
Content: Parts Stackforder Toge Parts Stackforder Toge Parts Stackforder Toge Parts Stackforder Toge Parts Totabert teacher Content Totabert teacher Content Totabert teacher Totabe
Content: Prise Stack determined and the second of the secon
Parts Stack-Control Stack-Control Stack-Control Trag Trag Trade there there Address Model Stack-Control Stack-Control Stack-Control Stack-Control Stack-Control Trade there there Stack-Control Sta
Constants
i Struchdar i Fifers i Modds i Wodds i Tubker tokorst Rato i Constant i Constant <t< th=""></t<>
Constant Model Mod
Contract Model Mod
Image: Section of the section of th
Below 1 Constant - Properties C
Reference Values Worker Controls Reference Values Refer
Prysca 1 Prysca 1 Prysca 1 Pressure Constant Constant Constant Constant Constant Constant Constant Constant Prysca 0 Constant Constant Prysca 0 Pressure Constant Constant Prysca 0
Source of the second with any region Source of the second with any re
Reformer Values R
Sortant Condens Contant Conta
Constant
Constant
Constant Co
Tublem Specification Tublem Specification Constant Co
Constant
Constant Organization Constant Con
Constant Constant Region 1 Constant Consta
Constant - Properties
Constant
Region 1 Cell type 0 not associated with any region Cell type 0 not associated with any region Cell type 0 not associated with any region (Cell type 0 n
Constant - Properties Constant -
Cell type 0 not associated with any region Constant - Properties III 0, 20, 00 mins III CK
Constant - Properties d # Constant - Properties d # Constant - Properties Constant - Pro
Properties CK.
Value [0 0, 2 0, 0 d] m's Unit - Star 2 (0 uptot
Output - Star 2
, DOUDOULOUT-CONTACT STATECHTY: [Litename not available] (Litename not available) [ERDRS YUGELLINEFronddarfd, GetLastError 487 (Adress 10000007678652D)
, 0000000/FF7652D (kernel32): (filename not available): BaeThreadInitThunk
, ERROR: SymGetLineFromAddr64, GetLastError: 487 (Address: 00000007731C521)
, 00000007731C521 (ntdl): (filename not available): RtUBerThreadStart
J error: Server Error
expression: r
Constant @ file: Mesh/KernelMesh.cpp
Constant vector profile line: 62
(2) (2) (2) (2) (2) (2) (2) (2) (2) (2)

4/21/11

Entry by: Cameron Hjeltness

Today I'm going to try working some more with the cylinder that has rounded edges.

I decide to look up the best mesh type to use:

- Thin mesher:
 - o Used to generate prismatic volume mesh for THIN regions
 - Two types:
 - Thin mesher
 - Default
 - Can be used in place of 1 or more of the following core volume meshers: polyhedral, tetrahedral, or trimmed cell.
 - Embedded Thin mesher
 - Very similar to the default thin mesher except this one assumes that its surrounded by another region that is part of the analysis.
 - Used alongside the polyhedral volume mesher
- Prism Layer Meshing:
 - o Composed of orthogonal prismatic cells
 - o Used for Turbulence and heat transfer analysis
 - Number of layers is determined by turbulence model used.

- For wall function based models: 1-3 layers
- For low Reynolds numbers/2-layer schemes 15-25 layers
- Its possible we weren't using enough layers for the 1st and 2nd runs
- Polyhedral Mesher
 - Used for complex situations
 - o About 5 times fewer cells than a tetrahedral mesh
 - Generalized cylinder mesher
 - Used in conjunction with the Polyhedral mesher
 - Looks like an aluminum honeycomb structure
 - Reduces the number of cells used
 - Improves rate of convergence
 - Select this option when the fluid flows parallel to the vessel wall, for example, pipe flow.
- Tetrahedral Mesher
 - o Also for complex situations
 - o Fastest model
 - Uses the least amount of memory

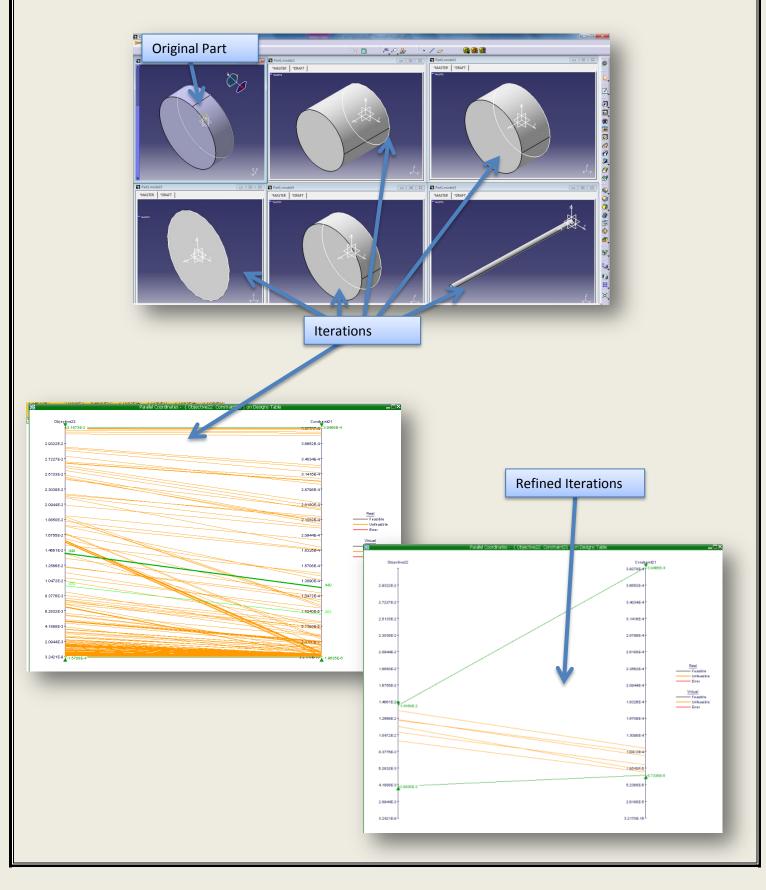
4/28/11

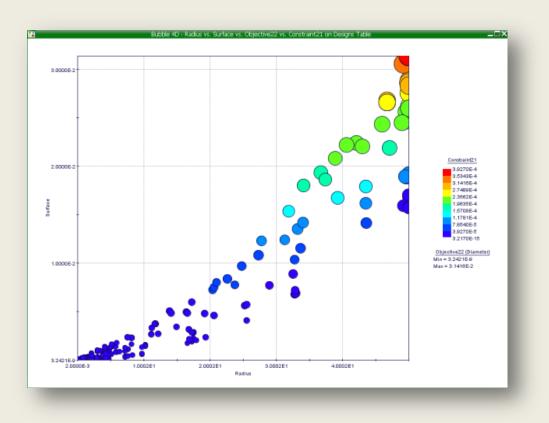
Entry by: Cameron Hjeltness

Today I met with Dr.'s Beyerlein and Schwarzkopf with the intent of finding a solution to running a 3D analysis with Star CCM+. Dr. Schwarzkopf has experience with CFD analysis and I was hoping he might be able to direct me toward a method of running the simulation. Unfortunately, the software we have is much more advanced than what he was used to in his field of study. One thing that we did notice however was that the problem seems to be within generating our volume mesh. He said that this is one of the harder parts, but once this is accomplished, the rest should be much easier to finish up.

4/28/11

Entry by: Cameron Hjeltness





These pictures demonstrate the optimization of a part in Mode Frontier. It starts with an original part and we provide a constraint such as this example where we constrained the volume and tried to minimize the surface area. Each of the bubbles represents an individual experiment that was run. As you can see, it follows a trend line representing the constrained volume, as the radius and length change in dimensions. The picture below the bubble chart represents the refining of the constraints. Say for instance, you know that you can't have a diameter over 1 meter, you would drag the left green arrow down and this effectively limits every design with a diameter over 1 meter. Through this process you then can select the best possible design.

For a more detailed description, please see next entry

5/2/11

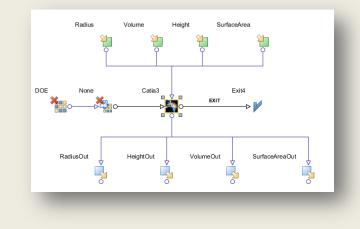
Entry by: Dylan Rinker

After much tedious effort I have finally figured out how to do basic optimization of a Catia part in modeFRONTIER. I will give a rundown of how to go about this from scratch.

- 1. Open modeFRONTIER.
- Every modeFRONTIER program must begin with a "Scheduler" node, so select the scheduler node and drag it into the workflow space. This node, when put into the workflow space, will appear as two nodes: a DOE node and Scheduler.
- 4. Every modeFRONTIER program must also end with a "Logic End" node 📝 , so drag that into the workflow (it is located in the drop down under the scheduler).
- 5. At this point the nodes can all be connected and it should look like this:



- 6. Next the CATIA file needs to be selected. Double click the CATIA node and CATIA Document Properties dialogue box will open. Under the Document option, browse for, and then select the part that you wish to optimize, and then click ok.
- Now it is time to select the inputs and outputs to your system. Drag an input node in for each of the inputs you would like to use, and connect them to your CATIA node. For every input node you need to also make an output node . You can name these accordingly and it should look like this:



8. Next go into the CATIA node again to connect each input and output node to a parameter in your part. To do this, double click on the CATIA node to open the CAITA Document Properties. At the bottom of this dialogue box should be the inputs on the left, and outputs on the right. Click the binoculars symbol in the blank field next to the input or output name. This will begin an introspection. Once the introspection is complete click close, and the design tree of your part will appear.

Catia	
E Length	
Part1	
Height	
Raduis	
Volume	
💕 Surface	
Surface Selected paran	neter Height
Surface Selected paran Parameter D Comment	neter Height
Selected param Parameter D Comment Hidden	neter Height Petails false
Selected paran Parameter D Comment Hidden Readonly	neter Height etails faise faise
Selected param Parameter D Comment Hidden	neter Height Petails false

Select the desired parameter to attach to the input/output from the design tree.

Double click the parameter in your design tree that you wish to link to each input and output. Do this for each input and output.

9. At this point you can also select the export and screen capture setting you want. In the CATIA Document Properties window you can expand Export option, and I suggest selecting "Model". This will export a CATIA model for each iteration of your optimization. Also, you can expand Screen Capture option and select to export an image with each iteration as well. Once the inputs, outputs, and export options are all set click ok.

CATIA Document Properties - 1.0 b20101110	X	
Edit CATIA Document	A rences	
CATIA Document Properties		
Name	Catia15	
Description		
Document	S:\HVLC\seniordesign\Catia Course\10. Spring 2	
Script File		
Is relative		
CATIA Document Advanced Properties		Select Export format
CATIA Work Space Properties		
Windows Only Properties		
Export		
Export as 3DMAP		
Export as CGR Export as IGS		Select Screen Capture format
Export as IGS Export as Model		
Export as Session		
Export as STL		
Export as STP		
Export as TXT		
Export as VPS0		
Export as WRL		
Screen capture		
Screen capture as CGM		
Screen capture as EMF		
Screen capture as TIFF		
Screen capture as TIFF GreyScale		
Screen capture as BMP		
Screen capture as JPEG		
		Dinequilars to access the design
Process Input Connector	Process Output Connector	Binoculars to access the design
Scheduler	📝 Exit20 🛛 EXIT 👻	
Data Input Connector	Data Output Connector	tree for each input and output
Height Height	🛍 🛃 Surface Surface Area 🏥	thee for each input and output
Radius Raduis	Volume Volume	
10000		
ОК	Cancel Help	

- 10. Now you need to set the boundary conditions for your inputs. Double click each input node and change the upper and lower bounds to narrow down the scope of your experiment and minimize infeasible results.
- 11. Similarly, set up an objective for each output. To do this, select the objective node and drag one under each output. Depending on your goal the objective can either be to minimize or maximize the output. Also, you may want to constrain an output, in which case you can drag a constraint node under the output (the constraint node is in the drop down of the objective node). Within the objective or constraint nodes you must select the "User Expression". To do this double click on an objective or constraint node and click on the little calculator symbol in the User Expression field. Select the variable you want, then click apply.

Objective Properties - 4.3.0 b20101110												
Objective Properties												
Name				Objective30_2								
Description						2						
Enabled									_			
Format	0.0000E0									Click on the little		
Objective Expression Properties												
User Expression	Node3axialOutput											calculator symbol.
Туре				Minimize							calculator symbol.	
Data Input Connector Jona Mode SaxialOutput												
ОК			G	ancel					Help			
Expression Editor	- * *							23	J			
Variables E	Expression											
										Expression	wir	Variable is in the dow, and is spelled
1	Basic Functi	ions			Ope	rators				exactly the	san	ne.
	sin	COS	tan	degToRad	~	&	!					
	asin	acos	atan	radToDeg]				
	log	In	exp	sqrt	% 7	, = 8 9		CA				
	abs	sgn	rand	pow	4	5 6		<-				
	ceil	floor	round	mod		2 3						
	min	max	interp	vect	0	E PI		End				
Apply					ancel		_					

12. Next you need to set your scheduler settings. Double click the DOE node. Under Space Fillers select Random. This is a good place to start with simple optimizations. Under parameters you can select the number of designs to start with. I picked 10.

DOE Properties - 4.3.0 b20101110		X	1	
1 🖄 🗲 😫 🗎 🍋 🗟				
Bace Fillers (8)	Random Design of experiments based on a range a uniform distribution, the design space me sequence of points is determined			Design of Experiment (DOE)
Kandom Sobol Sobol Uniform Latin Hypercube Incremental Space Filer Constraint Satisfaction Robustness and Reliability Latin Hypercube - Monte Taguchi Orthogonal Arrays Statistical Designs	The equance of ponts is determined Three parameters can be defined: 1) Number of experiments to be ger 2) Reject or accept unfeasible design 3) Random seed for sequence repea It can be used as initial design populat The number of generated designs is li Parameters Number of Designs [1,256000] Reject Unfeasible Samples Random Generator Seed [0,999]	rerated; is; izoliity. ion for MOGA and Simplex algorithm. mited to 256000.		Number of initial designs
Full Factorial Reduced Factorial	Add DOE Sequence	Stop DOE Sequence		Number of initial designs
DOE Designs Table DOE Log M CATEGORY Theight 0 RNDDOE 4.6176E2 1 RNDDOE 5.8457E2 2 RNDDOE 9.3551E2 3 RNDDOE 9.2741E2 4 RNDDOE 9.4939E2 5 RNDDOE -2.0565E2	Radius -1.7994E2 -3.3457E2 -9.8777E2 8.9735E2 8.7416E2 -3.0496E2			
N. Designs:10	N. Error Des.:0	N. UnFeasible Des.:0		
ОК	Cancel	Help		

Once these two options are set, click Add DOE Sequence, and this will tabulate the initial designs for your experiment, then click ok.

13. Double click on the scheduler node that is connected to the DOE node and pick which algorithm

you would like to use for your optimization. I picked MOGA-II.

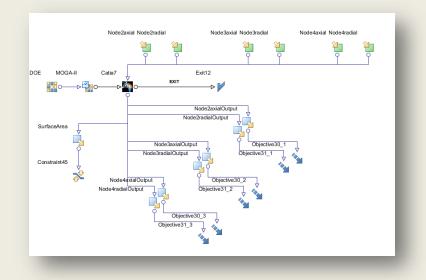
Optimization Wizard	🗌 🗆 MOGA-II 🛛 🔯 🛐 🖆	
Schedulers	Scheduler based on Multi Objective Genetic Algorithm (MOGA) designed for fast Pareto convergence.	
DOE Sequence		
MACK	Main features:	
Lipschitz Sampling	 Supports geographical selection and directional cross-over. Implements Elitism for multiobjective search. 	
	3) Enforces user defined constraints by objective function penalization.	
Basic Optimizers	 4) Allows Generational or Steady State evolution. 5) Allows concurrent evaluation of independent individuals. 	
SIMPLEX		
B-BFGS	The N (num. of individuals) entries in the DOE table are used as the problem's initial population. Each input variable base must be different	Select your desired genetic algorith
Levenberg-Marquardt	from zero, since MOCA II works only with discrete variables.	
🕷 MOGA-II	Parameters	
ARMOGA	Number of Generations [1,5000] 50	
	Probability of Directional[0.0,1.0] 0.5	
🚭 Advanced Optimizers	Probability of Selection [0.0,1.0] 0.05	
NSGA-II	Probability of Mutation [0.0,1.0] 0.1	
MOSA	Advanced Parameters	
MOGT	DNA String Mutation Ratio [0.0,1.0] 0.05	
	Elitism Enabled 🗸	
MOPSO	Treat Constraints Penalising Objectives	
FMOGA-II	Algorithm Type MOGA - Generational Evolution	
FSIMPLEX	Random Generator Seed [0,999] 1	
Evolution Strategies	Category Parameters	
www.Evolution Stratedies		
Run Options RSM Options MOR		
Run Options		
Num. of Concurrent Design Evalu	uations 1	
Save Error Design in DB		
Evaluate Repeated Designs		
Save Repeated Design in DB		
Evaluate Unfeasible Designs	V	
Clear Design Dir on Exit	Never	
OK	Cancel Help	

15. The experiment may take a while to run, but after it is complete you can use the information collected to choose an optimum iteration. To do this click on the Designs Space tab. In the Design Charts section of Tables and Charts click on Parallel Coordinate. Select the objectives that you would like to base this optimization on, and then it will bring up a graph consisting of a bunch of lines with a few vertical axes. Each vertical axis has objectives on it. Sliding the green arrow up and down the axes will eliminate or include different iterations. After narrowing down which iteration best fits your desired results you can select it and view the exported model, screen shot, and also run the exported CATScript to have CATIA automatically change your part to that iteration.

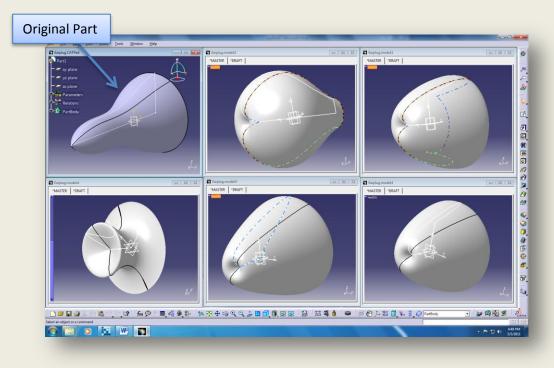
5/3/11

Entry by: Dylan Rinker

I preformed another more complex optimization of an earplug shaped part. Here is the workflow of the modeFRONTIER program:



In retrospect, this experiment's objectives were poorly defined and resulted in some amusing, but irrational shapes, as far as aerodynamics is concerned. Now that we have the process down for doing optimization, the really hard part will be creating a workflow program to run an experiment which will achieve applicable results. Here are some iterations of the interesting, yet inapplicable results of this experiment:



5/3/11

Entry by: Cameron Hjeltness

How to creat and define a geometry within Star CCM+

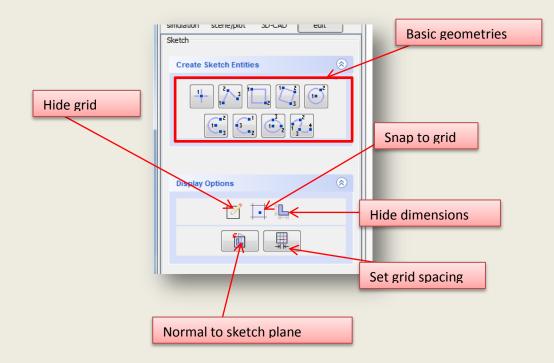
File>>New Simulation.

Accept basic configurations.

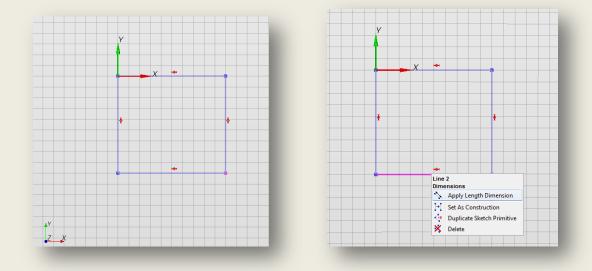
Expand Geometry, right click on 3D CAD Models and click create new.

Note: More than likely, you will simply import an existing model from CATIA or Solidworks. To do this, just go File>>Import>>Volume Mesh. The following is information on how to create geometries within Star CCM+.

Right click on one of the planes, and click creat ew sketch. From here you can create geometries in the familiar fashion that you are used to from Solid Works or CATIA. The only difference is that if you want to want to create something new, more often than not, you have to right click to open up a function.



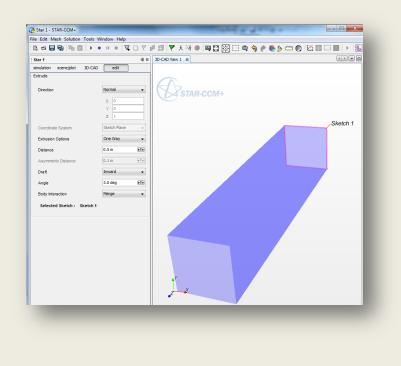
For demonstrational purposes I will generate a simple part with an angled extrude.



First create the profile. If you want to arbitrarilly redfine the size of the shape, just click and drag. Alternatively, right click and set the dimensions.

Right click on the newly created sketch and create extrude. Note all of the other possibilities in the same menu.

To create another feature on this part like an extrude cut, click on the face you wish to start the sketch from and create a new sketch.

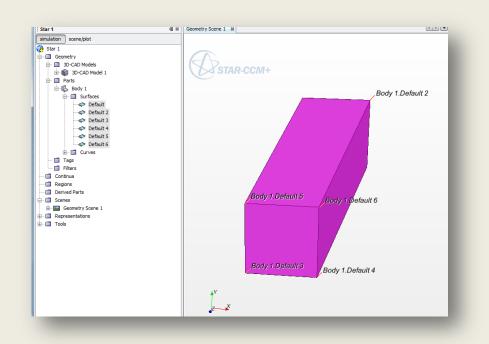


Click OK. Close the 3D-CAD.

Right click 3D-CAD Model1 and select new geometry part. To view this new part, RC (right click) scenes and select new geometry scene. This is where all visualizations will be located. This includes scalar, vector, geometry, mesh and empty scenes.

Expand "Surfaces" and RC on the default surface. Currently there is only one part surface. We want to break this up so that we can define things like the wall and the inlet/outlet ports. Select Split by angle. Define the angle as 45 degrees. This means that any angle that Star CCM detects that is over this value, it will create a new boundary at that point.

You should now have 6 surfaces.



RC regions >> create new.

RC Body 1 >> Set region >> Region 1

RC Boundaries and then create 2 more boundaries. We need 3 total. 1 for the inlet, 1 for the outlet, and 1 for the wall constraining the fluid.

RC Boundary 1, then click the button with 3 dots. Select the 1st, 3rd, 4th, and 5th surfaces. To ensure that you are selecting the correct surfces, click on the individal surfaces and they will highlight in the geometry scene. The objective here is to create the wall for our boundary.

The default selection for this is a wall, which is what we want.

Star 1	41 X	Geometry Scene 1 28	
simulation scene/plot			
🕜 Edit		X	
@ Expand/Contract Tree @ Expand	l/Contract Values		
Nodes	Values	CM+	
🖃 🚃 Boundary 1			
Index	2		
Туре	Wall	· ·	
Interfaces Part Surfaces		ult 4, Body 1.Default 5,	
		Region 1:	Doundon: 1
Boundary 1			Soundary 1
Search by Name P			Soundary 1
_		t View III List View Alf Alf 2 Alf 4 Alf 5 S	Soundary 1
Show All Vane P		steven III Las Venn Net Net 2 Net 3 Net 4 Net 4 Net 4 Net 5 Net 6	Soundary 1
Search by Name P . Show All V		steven III Las Venn Net Net 2 Net 3 Net 4 Net 4 Net 4 Net 5 Net 6	Soundary 1

Boudary 2 >> Properties >> select the 2nd surface.

In the properties, make this a velocity inlet.

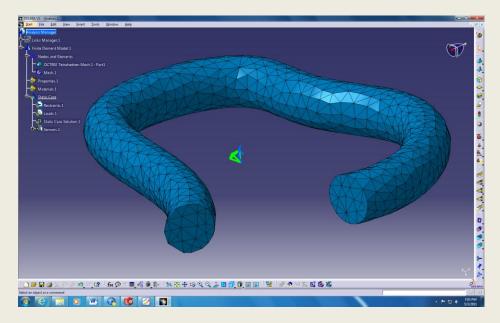
Default >> Properties >> Select the 6th surface.

In the properties, make this a pressure outlet.

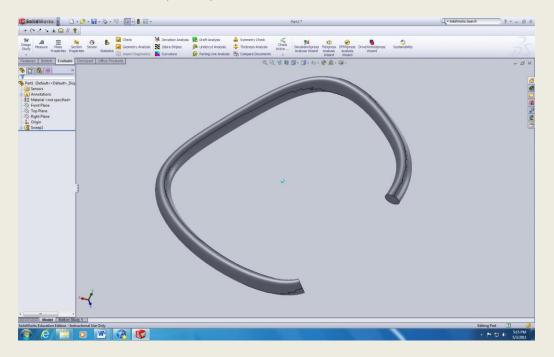
5/3/11

Entry by: Justin Dilworth

I thought the volume mesh was the issue so tried to see if there is a way to import catia's mesh making into a Star CCM+. I discovered that there is no way to import the mesh that catia makes into Star CCM+.



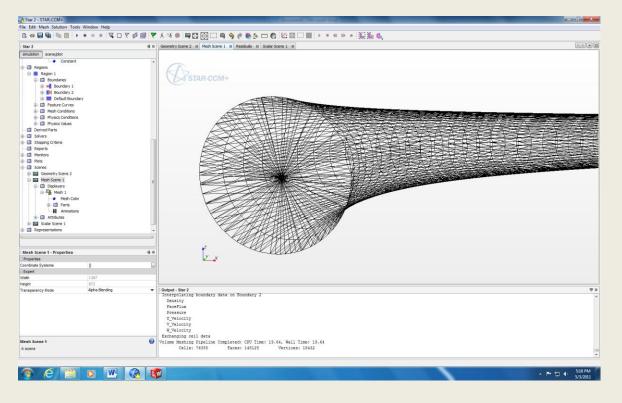
I made a part in Solidworks and saved it as a parasolid to import into Star CCM+. This created a curve in so that I could break the part imported into different surfaces.



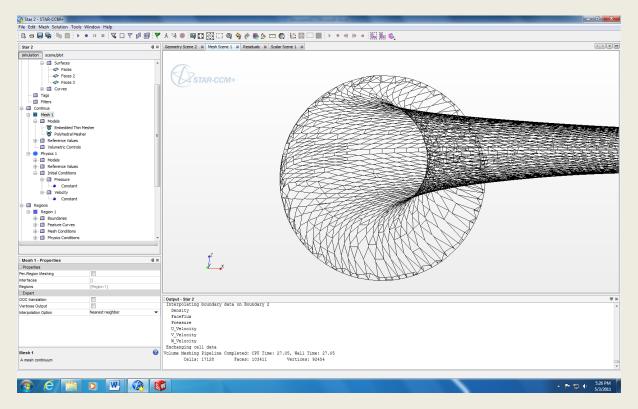
●御殿をします。◆●●「田田間のの日のの中での「田田間のの日のの」を「田田間のの日間」。◆●●	Star 2 - STAR-CCM+	
Image:	File Edit Mesh Solution Tools Window Help	
National State Image: State Image: State Image: State Image: State Image: State Image: State Image: State Image: State Image: State Image: State Image: State Image: State Image: State <td>D, es 💭 🗞 🗠 🖄 🔸 🔹 🗉 🥰 D</td> <td> ▼ 大 3 ● ■鼠園 亀 💁 使 巻身 🗂 動 2 副 ▶ = 4 ▶ + 副論 🗞</td>	D, es 💭 🗞 🗠 🖄 🔸 🔹 🗉 🥰 D	▼ 大 3 ● ■鼠園 亀 💁 使 巻身 🗂 動 2 副 ▶ = 4 ▶ + 副論 🗞
Indexest Indexest index Indexest Index		R II Geometry Scene 2 II Mesh Scene 1 II Residuals II Scalar Scene 1 II
Image: mode Image: mode </th <td>simulation scene/plot</td> <td></td>	simulation scene/plot	
Image: construction Image: construction Image: construction Image: construction <td>Reference Values</td> <td></td>	Reference Values	
Image: constrained in the second of the s	Initial Conditions	
Image: work Image: work <td></td> <td></td>		
• Constant • Norma		CASIAR-CCIN+
Image: Statistic Statisti		
Image: State Cores Image: State Cores Image: State Cores Image: State State Cores Image: S		
Image: standard i intervente intervent		
Web Keet 1 Web Keet 1		
Image: Source is a section of the		
Image: Source Source Image: Source Source Image: Source Source Source Image: Source Source Source Image: Source Source Source Image: Source Source Source Image: Source Image: Source Source Source Source		
Image: Second secon		
Image: Source		
Impact outling Impac		MULHAAA TAA KA K
Image: Second		NUMAN INVE V VIVIA AT I SKIMM
Barry One Barry One Construction of the C		
Avoid Server 1 Avoid Se	Derived Parts	
Notis Image: State of the s	8- 🖬 Solvers	
Notice 1		
Source for a constraint of the constraint o		
Construction C		
Converty fore 2 C		
Web Kones I Importante Board Properties Imp		
Agreent Source 1 Agreent Agree		
Tome Terretures Tome		
Tonic Ind		
Mask Scene 1 0 Mask Scene 1 0 Mask Scene 1 0 Mask Scene 1 0		
Conclusion Systems 0 Extent With Maget 027 Transporting Mode Apha Serving Compared, Sar 2 Extent Serving Compared, Sar 2 Description Compare	i ious	
Conclusion Systems 0 Extent With Maget 027 Transporting Mode Apha Serving Compared, Sar 2 Extent Serving Compared, Sar 2 Description Compare		
Concluse System 0 Deset 0 wmm 027 mapt 027 Tanaparency liste Apha Bendry Exercise Compared, Sar P Description Compared, Sar P		
Source mage: mag: mage:		
Vector 107 Medit 072 Temparency Mode AdpharBendig AdpharBendig 7 Temparency Mode AdpharBendig Vector 7 Temparency Mode AdpharBendig Vector 7		
Image megat 072 Image megat 072 Transparency Mode Adrie Binning • [Outget-ther 2 Transparency Mode Transpar		
Temperatory Note Adpts Broding Topological Bar 2 Temperatory Note Adpts Broding Temperatory Note Adpts Broding Temperatory Note Temperatory Note Mask Second 1 Optimic Adpts of temperatory Note Temperatory Note Mask Second 1 Optimic Adpts of temperatory Note Temperatory Note		
Texterplating buildary data in Booddary 2 Denity FaceTime Prestrum Prestrum ViPacity ViPacity ViPacity ViPacity ViPacity Excharging cull data Excharging cull data		Comput-Star 2
Benisty Particle Particle Particle UNaction UNaction UNaction UNaction Wightform Unaction Wightfor <td>Transparency Mode Alpha blending</td> <td> Uduput-Sker 2 Intercolation boundary data on Boundary 2 </td>	Transparency Mode Alpha blending	 Uduput-Sker 2 Intercolation boundary data on Boundary 2
Beak Scene 1 Øresarre 0 Vjekacity Vjekacity Vjekacity Vjekacity Exchanging cell data Exchanging cell data		
Unitedity Unitedity Wash Newes 1 Windows projection completed CPD Times 19.44, Wall Times 19.44		FaceFlux
V_Welscity Welscity Welscity Exchanging cell data Kenne Y V		
WiveLocity Exchanging cell data Weak Scene 1 V Vive Menhing Jupilion Completed: CNV Time: 19.64, Wall Time: 19.64		
Meah Sense 1 Volume Menhang Fysician Completedis CFU Time: 19.44, Wall Time: 19.44		
Meah Scene 1 🚱 Volume Meshing Fipeline Completed: CRU Time: 19.64, Wall Time: 19.64		
	Mesh Scene 1	
	🔊 🤶 🐃 o 🕅	SW - > 33590

This is the surface mesh the import part from solid works created.

After splitting up the surfaces into proper boundaries I then had it try and make a volume mesh, and putting into wireframe mode, it appears the volume mesh isn't properly made and is hollow.



Switch over to a polyhedral mesh, and this confirmed that the volume meshes generated are hollow, and this could be why we are not getting any diagrams when running our simulation.



On a side note from the volume generation errors generated, I found a way to clear the residuals that you have collected so that you can start over as needed. Go to the eraser icon located as shown.

🙀 Star 2 - STAR-CCM+										
File Edit Mesh	Solut	tion Tools Windo	w Help							
R 🗠 🛢 🛃	0	Clear Solution		7		$\overline{\nabla}$	00	1		
Star 2	7	Initialize Solution		F				411 88		
	÷\$	Run	Ctrl+R	┝				NI 60		
	¥	Step	Ctrl+T	┝				_		
Tags	R	Step Control						^		
🚊 🖬 Continua										
🖨 🛄 Mesh 1	🖶 🗰 Mesh 1									
🕀 🖷 Models										
🕀 🗂 Reference Values										
Volumetric Controls										
🚊 🤤 Physics	🖻 🐡 🧇 Physics 1									
📙 📩 🖬 Mo	dale									

I believe that the reason why we can't get results is from not getting our volume meshes to work. When trying to create a surface mesh it appears there is a bug in our software preventing it from doing it right.