# Individual task Objective

To derive the drag coefficient for a 2D object, defined as  $C_D = \frac{2D}{\rho U^2 H}$  where D (N/m) is the aerodynamic drag force (per unit length in the third direction) acting on the object. The object could be one of the following (or an object of your choice).



## Settings

- Choose height, *H*, free stream velocity, *U*, and fluid (dynamic viscosity  $\mu$  and density  $\rho$ ) so that:
- Reynolds number  $Re = \frac{\rho UH}{\mu}$  (choose 10<sup>4</sup>, 10<sup>5</sup> or 10<sup>6</sup>)
- Mach number  $Ma = \frac{U}{a} < 0.1$  (where *a* is speed of sound)
- Observe, the size of the object *H* is the projected size in the streamwise direction (see figure).

## What to do (read carefully)

- Preparation for lecture 2 (31/3): Make a sketch of your case and the estimated flow field around it. What physical model (inviscid, laminar, turbulent, compressible) should be used? What is the Reynolds number? Make a sketch of the computational mesh.
- During (and after) lecture 2 (31/3): Determine the grid resolution requirements. What is a suitable near-wall grid size (thickness of the first grid cell) in the wall normal direction considering  $y^+$  requirements for a log-law boundary conditions ( $y^+$  =30-100) or a fully resolved boundary layer ( $y^+$  =1)?
- Before 18/4 23:59:
  - Run the case using Fluent for obtaining the drag coefficient following the step-by-step instruction. It is recommended that you do this before the Fluent tutorial 13-14 April.
  - Create two figures, one with the grid and one with the velocity field, and give the drag coefficient. Upload the results on bilda. Post your results in the Discussion Forum "Individual Task" (see the example by me).
- Feedback: Lecture 5, 20/4

## **Step-by-step instruction**

Follow the detailed step-by-step instruction at the course web: <u>https://www.kth.se/social/page/course-documents</u>: Individual Task

# How to setup this case in Fluent – Workbench

Start ANSYS Workbench: All Programs -> Ansys 16.0 -> Workbench

(All figures here are from Ansys 13.0 and there are some differences compared with the current version 16.0 or 16.2. Try to use the most appropriate when differences).



Setup a new computation

- 1. Drag and drop "Fluid Flow (FLUENT)" from Toolbox to Project Schematic
- 2. Set a name in the box
- 3. Save the project: File -> Save As...

Documentation: Help -> ANSYS Workbench Help -> Workbench -> ANSYS Workbench User's Guide -> Getting Started in ANSYS Workbench -> The ANSYS Workbench Interface

## How to create the geometry – DesignModeler

Start DesignModeler: DoubleClick on "Geometry" in the Workbench Scheme

😳 C: Geometry - DesignModeler		
File Create Concept Tools View Help		
🛛 🛃 📕 👛 🗍 🏵 Undo 📿 Redo	Select: 🍾 🏷 🖻 🛅 🕞 🚱 😓 🔆 🔍 🔍 🍭 🔍 🔍 💥 🤸 📦 🕫	
XYPlane 💌 🛧 None 🕤	- 29	
📙 誟 Generate 🛛 🖤 Share Topology 🕴 💽 Ext	trude 🚓 Revolve 🐁 Sweep 🚯 Skin/Loft 📗 Thin/Surface 🦠 Blend - 🦠 Chamfer 🚸 Point 🔀 Parameters	
Tree Outline 7	Graphics	ą
<ul> <li>→ Geometry</li> <li>→ XYPlane</li> <li>→ XZPlane</li> <li>→ YZPlane</li> <li>→ YZPlane</li> <li>→ 0 Parts, 0 Bodies</li> </ul>	3	*
Sketching Modeling Details View 4	2	
	0,00 <u>30,00 (m)</u>	×
	Model View Print Preview	
🥝 Ready	No Selection Meter 0 (	0 //

- 1. Select length unit: Meter
- 2. To see a x-y view: Click on the "z" coordinate.
- 3. To work in the x-y plane: Click on "XYPlane"

#### Sketch your geometry

🚥 B: Geometry - DesignModeler							X
File Create Concept Tools View Help							
🖉 📕 📕 🖚 🔊 Undo @ Redo Se	elect: *💫 🖏 🕅	n 🛯 🖓 - I S 🕸		2 🚟 🖈 🚳 🕫	2		
XVPlane	a						
	Annulus A Course	A clash of marketing	an Antonio Archard				
	Burkevolve @ Sweep	Skin/Lorc (Chin/Surra		er wont pspara	meters		
Sketching Toolboxes 4 Grap	phics			~		_	4
Draw						70	
Line	6				VIV S	YS	
6 Tangent Line	V						
6 Line by 2 Tangents							
APolyline							
Polygon							
Pectangle by 3 Points							
2 Oval							
Modify 👻							
Dimensions		1					
Constraints	5						
Salling	Y						
Secongs							
Sketching Modeling							
Details View 4							
Details of Sketch1			1			v	
Sketch Sketch1							
Sketch Visibility Show Sketch							
Show Constraints? No		0.000		5 000 (m)		┟ →	X
Edges: 6		0,000		5,000 (m)			
Circular Arc Cr7			2,500				
Line Ln8							_
Line Ln9 Mo	del View Print Preview						
Rectangle Click, or Press and Hold, for first co	corner of rectangle		No Selection		Meter	-2 4,	5

- 4. To work in the Sketching mode: Click on "Sketching"
- 5. Sketching Toolboxes -> Settings ->
  - a. Grid: Show and Snap on
  - b. Major Grid Spacing: 1m

- c. Minor-Step per Major: 5
- d. Snaps per Minor: 2
- 6. Sketching Toolboxes -> Draw ->
  - a. Arc by Center: Click on origo, first and second arc points (radius = 0.5)
  - b. Line: Click on first and second arc points.
  - c. Rectangle: Drag a rectangle x from -4 to 10 and y from -4 to 4.

Create surface (computational domain)



- 7. Concept -> Surfaces from Edges ->
  - a. Concept -> Surfaces from Sketches ->
  - b. Select all edges <CTR> Click
  - c. Click Generate
- 8. Close "DesignModeler"

Now, the Geometry cell in the Workbench Scheme should be checked green.

Documentation: Help -> ANSYS Workbench Help -> DesignModeler ->

## How to create the mesh –Meshing

Start Mesher: DoubleClick on "Mesh" in the Workbench Scheme



- 1. Make a first mesh
  - a. Click on Mesh
  - b. Set Sizing -> Relevance Center to "Fine"
  - c. Generate Mesh: Mesh -> Generate Mesh

🕲 B : Mesh - Meshing	[ANSYS Academic	Research CFD]				
File Edit View Units	Tools Help	👪 \Lambda 🞯 - 🌒 Worksheet				
🛒 <sup>K,K,Z</sup> 🔓 💽	R R R 🚱 -		Q Q 🐺 /0 🗃 🗌	]-		
Mesh ジ Update 🏻 🌚	Mes - 🔍 Mesh Contro	ol •Metric Graph Q Opt	tions			
Outline		4				
Project Pro	e Systems Sizing 2b	Edge Sizing 2010,02,18 14:44 25 Edg Sizing		20	Noncommercial	SYS use only
Details of "Edge Staing" - Siz	zing	<del>.</del>				
Scope						
Scoping Method	Geometry Selection	2d				Y
Geometry	2 Edges	2u				1.
Definition						<b>X</b>
Suppressed	No		0,000	A REALING	0,800 (m)	
Туре	Number of Divisions	2e		0,400		
Number of Divisions	40					
Behavior	Soft	INI semistry your	t Preview A Report Preview/			
Curvature Normal Angle	Default	Me sage				Ψ×
Growth Rate	Default	Text			Asso	ciation
Bias Type						
Bias Factor	20,	20				
		22 2h				
Press F1 for Help		🟓 No I	Messages No Selection		Metric (m, kg, N, s, V, A) C	egrees rad/s

- 2. Refine mesh around half sphere
  - a. Click on Mesh Control -> Sizing
  - b. Click on edge symbol
  - c. Select the half sphere: <CTR> Click on arc and line
  - d. Click on Geometry -> Apply
  - e. Set Type = Number of Divisions
  - f. Set Number of Divisions = 40
  - g. Set Bias Type = ".-\_-."
  - h. Set Bias Factor = 10 (20 as in the figure does not work)
  - i. Generate Mesh: Mesh -> Generate Mesh
  - j. Click on Mesh to see it



🕲 B : Mesh - Mes	shing [ANSYS Academic	Research CFD]		
File Edit View L	Jnits Tools Help	🚯 \Lambda 🐼 - 💕 Worksheet		
			150 (0 B	
1				
Mesh 😏 Update	Mesh • • Mesh Contr	al - all Metric Graph   Coptions		
Outline				
Project ⊖ @ Model (8 ⊖ ¬♥ Geor ⊖ ↓ Coor ⊖ ↓ © Mest	2) netry drifte Systems E oge storing Innacion	3b	31	Noncommercial use only
Details of "Inflation" -	Inflation	4		
🖃 Scope				
Scoping Method	Geometry Selection	3d		
Geometry	1 Face		20	
Definition			30	
Suppressed	No	20	0,000	0,800 (m)
Boundary	2 Edges	32	0,400	
Inflation Option	First Layer Thickness		N=	
First Layer Height	1,e-003 m	Bometry Annufrey BW	Keport Preview/	
Maximum Layers	20	Me sages 1		4 ×
Growth Rate	1,2	Text		Association
Press F1 for Help		No Messages	No Selection	Metric (m, kg, N, s, V, A) Degrees rad/s

- 3. Make a boundary layer mesh around the half sphere
  - a. Click on Mesh Control -> Inflation
  - b. Click on face symbol
  - c. Select the computational domain
  - d. Click on Geometry -> Apply
  - e. Click on edge symbol
  - f. Select the half sphere: <CTR> Click on arc and line
  - g. Click on Boundary -> Apply
  - h. Inflation Option = First Layer Thickness
  - i. First Layer Height = 0,001 m
  - j. Maximum Layers = 20
  - k. Generate Mesh: Mesh -> Generate Mesh
  - l. Click on Mesh to see it

🕲 B : Mesh - Meshing	[ANSYS Academic Resea	rch CFD] 📃 🗖 🔀
File Edit View Units	Tools Help 📗 🏥 👪	A) 🕢 🗸 🕼 Worksheet
🛛 📽 🦹 🔓 🕞 🕅	C - 🖗 🗊 🗊	∲ Q ⊕   Q Q Q \$\$ 12 12 ⊨   □ -
Mesh 岁 Update 🏻 🌚 I	Mesh 👻 🔍 Mesh Control 👻	Il Metric Graph Options
Outline Project Model (82) Profit Geometry Coordinate Mesh Details of "Mesh"	: Systems Sizing ion	
<ul> <li>Defaults</li> </ul>		
Physics Preference	CFD	
Solver Preference	Fluent	
Relevance	0	
Sizing		LA 2000 X X LA COMPANY
Use Advanced Size Fund	tion On: Proximity and Curv	
Relevance Center	Fine	
Initial Size Seed	Active Assembly	\Geometry APrint Preview A Report Preview /

B : Mesh - Meshing [ANSYS Academic	Research CFD]		
File Edit View Units Tools Help	👪 🖪 🕖 - 💕 Worksheet		
🐨 👯 🔓 - 🔁 💽 💽 🥰 -	5 ÷ @ @ @ Q Q	🞇 /0, 🖴 🔲 -	
Model   II Virtual Topology	😌 connections 🛛 🕸 Mesh Numbering 📗		
Outline       2         Image: Superson of the second s	A Anned Selections 2010-03-18 15:07 A Inlet B Outlet C Symmetry D Obj		Concommercial use only
Details of "Named Selections"	4c,	0.000	6.000 (m)
Display		3,000	
Show Mesh Yes	Commenter (Drink Demainer)	Discount Description	
		Report Preview	
	Messages		Association
Press F1 for Help	0 No Messages	No Selection	Metric (m, kg, N, s, V, A) Degrees rad/s

- 4. Set name on the boundaries
  - a. Click on Geometry
  - b. Click on edge symbol
  - c. Select the inlet boundary
  - d. Right-click on the selected boundary -> Create Named Selection
  - e. Give boundary name
  - f. Do this for "Inlet", "Outlet", "Upper", "Lower" and "Object". Use <CTR> click for choosing multiple boundaries. Names are arbitrary.

File Edit View Units Tools Help     Image: State Edit     Image: State	🕲 B : Mesh - Mes	hing [ANSYS Academic Rese	arch CFD]			_ 🗆 🔀
Better in the interview     Operation     Better interview     Operation     State Body     State Body     Definition     De	File Edit View U	Inits Tools Help	A 🕢 - 💕 Worksheet			
Geometry Cache Project Project Segmetry S	P		4 ® A   ® 0		1.	
Geometry       Project         Project       Surface Body         Surface Body       Salface Body         Suppressed       No         Coordnate System       Salface Body         Suppressed       No         Coordnate System       Salface Body         Suppressed       No         Coordnate System       Hesages         Text       Association         Thickness Mode       Manual         Offset Type       Mode(Membrane)         Sald       Sold         Properties       Sold         Statistics       No Messages         No       Sold         Statistics       No Messages	4 4 43 · U		1 4 4 4 A		_ **	
Outline       P         Project       P         P Model (B2)       Sufface Body         P Coordinate System       Sufface Body         P Graphics Properties       Sages         P Fraphics Properties       Sages         P Trickness Mode       Material         Thickness       Model (Membrane)         Suppreties       Suppreties         Properties       Suppreties         Properties       Suppreties         Properties       Sud         Properties       Sud         Properties       Sud         Properties       Sud	Geometry					
Project     Sufface Body     Suffac	Outline		<del>•</del>			
Image: Surface Body       Surface Body         Image: Surface Body       Summery         Image: Surface Body       Surface Body         Image: Surface Body       Sufface Body         Image: Surface Body       Sufface Body         Image: Surface Body       Sufface Body         Image: Sufface Body       Sufface Body	Project					
Image: Second Properties       58         Image: Second Properties       5000 model         Image: Second Properties       500 model	E- Model (B2	2)				
Image: Statistics	- Geom	Surface Rodu			Noncommerci	al use only
Details of "Surface Body"   © Named Selections   © Named Selections   © Symmetry   © Object     Definition   Suppressed   No   Coordnate System   Default Coordnate System   Thickness   0.00   6,000 (m)   3,000     Period     No   Coordnate System   Default Coordnate System     Thickness   0.01   Suppressed     No   Coordnate System   Default Coordnate System     Thickness   0.01   Suppressed   No   Coordnate System   Default Coordnate System   Thickness   0.01   Suppressed   No   Coordnate System   Thickness   0.02   Suppressed   No   Messages   Thickness   Solid   Bounding Box   Properties   Statistics   No Messages No Selection	F coor	dinate Systems				
Statistics     Statistics	- On Mesh					
Inflation   Image: System of Subrace Body   Outlet   Symmetry   Object     Details of "Surface Body"     Operation of "Surface Body"     Outlet     Operation of "Surface Body"     Operation of "Surface Body"     Operation of "Surface Body"     Outlet     Statistics     Properties     Statistics     Properties     Onto Messages     No Selection     Metric (m. ko. N. s. V. A)     Decision of the properties     Properties     Statistics     Properties     Onto Messages     No Selection     Metric (m. ko. N. s. V. A)	, @,	Edge Sizing	5a			
Cordinate System     Details of "Surface Body"     Outlet     Symmetry     Outlet     Outlet     Outlet     Outlet     Outlet     Outlet     Outl	1	Inflation		A		
Coordinate System     Defailt Coordinate System     Thickness     Mo     M	E- 🐨 Name	ed Selections				
Outlet       Outlet         Symmetry       Symmetry         Statistics       0,000         Supressed       No         Coordinate System       Coordinate System         Thickness       0,001         Offset Type       Middle(Membrane)         Statistics       Statistics         Properties       Statistics		Inlet				
Object       0,000       6,000 (m)         © Graphics Properties       0,000       6,000 (m)         © Definition       0,000       0,000         © Supressed       No       Coordinate System       Thickness         Thickness       0,001       Messages       4 ×         Thickness       0,001       Text       Association         Offset Type       Middle(Membrane)       50       Text       Association         Properties       9       Statistics       Properties       9       No Messages       Metric (m, ko, N, s, V, A)       Denses       radia (metric)		Outlet				
Details of "Surface Body"     0,000     6,000 (m)       © Definition     3,000     3,000       © Suppressed     No       Coordinate System     Viscometry / Print Preview / Report Preview /       Coordinate System     Messages       Thickness     0,01       Thickness     0,01       Thickness     0,01       Thickness     0,01       Thickness     0,01       Thickness     Association       Offset Type     Middle(Membrane)       50     Fluid/Solid       Bounding Box     Properties       9     Statistics		Object				Y
Details of "Surface Body"     0,000     6,000 (m)       Graphics Properties     3,000       Definition     3,000       Suppressed     No       Coordinate System     Default Coordinate System       Thickness     0,001       Messages     # ×       Association     Association       Offset Type     Middle(Membrane)       Solid     Bounding Box       Properties     Statistics	~	object				t .
October of Sulface Gody     0,000     6,000 (m)       © Craphics Properties     3,000       © Suppressed     No       Coordinate System     Default Coordinate System       Thickness     0,00       Messages     4 x       Thickness     0,01       Thickness     4 x       Association     Association       Offset Type     Middle(Membrane)       50     Fluid/Solid       © Bounding Box     Properties       © Statistics     Yness FL for Help	Details of "Surface Roy	4.2				Z . X
Definition     Statistics     S	+ Graphics Proper	ties		0,000	6,000 (m)	
Suppressed No Coordinate System Default Coordinate System Thickness 0,01 Text Messages Text Association Offset Type Middle(Membrane) Material Fluid/Solid Bounding Box Properties Statistics Text No Selection Metric (m, ko, N, s, Y, A) Decrees rad/s	Definition		5a –		3,000	
Coordinate System Default Coordinate System Thickness 0.001 4 × 0.	Suppressed	No	A Geometry Print	Preview & Report Preview /		
Thickness     0,01     Messages     4 ×       Thickness Mode     Manual     Text     Association       Offset Type     Middle(Membrane)     50       Material     Fluid/Sold     Solid       Phoperties     Statistics	Coordinate System	Default Coordinate System	Indedined y/(i fille	TOTOM/(Toporerrottom)		
Thickness Mode Manual (Association)  Offset Type Middle(Membrane)  Material  Fluid/Solid  Bounding Box  Properties  Statistics  Tress F1 for Help  O No Messages No Selection  Metric (m, ko, N, s, Y, A) Degrees rad/s	Thickness	0,01	Messages			4 x
Offset Type     Middle(Membrane)       Material     5b       Fluid/Solid     Solid       Bounding Box     Properties       Statistics     Tress F1 for Help	Thickness Mode	Manual	Text			Association
Material  Fluid/Solid  Bounding Box  Properties  Statistics  Tress F1 for Help  Prose F1 for Help  Procest F1 fo	Offset Type	Middle(Membrane)	5h			
Fluid/Solid     Solid       Bounding Box       Properties       Statistics         Yress F1 for Help         O No Messages   No Selection   Metric (m, ko, N, s, Y, A) Degrees rad/s	- Material		<b>SN</b>			
Bounding Box Properties Statistics Tress F1 for Help O No Messages No Selection Metric (m, ko, N, s, V, A) Degrees rad/s	Fluid/Solid	Solid				
Properties     Statistics  ress F1 for Help      Metric (m, ko, N, s, V, A) Decrees rad/s	🛨 Bounding Box	Research Con-				
Statistics      Tess F1 for Help      O No Messages     No Selection     Metric (m, ko, N, s, V, A)     Deprees rad/s	Properties					
Press F1 for Help Metric (m. ko, N. s. V. A) Degrees rad/s	Statistics					
PIQUE UII NU 19 2 Y AL LOUICES 100 2	Press E1 for Help		0 No M	essages No Selection	Metric (m. ko. N. s. V. J	A) Degrees rad/s

- 5. Set thickness. The mesh must be 3D, but could be a thin slice that will be transformed back to 2D in Fluent.
  - a. Click on Geometry -> Surface Body
  - b. Set thickness to e.g. 0,01
- 6. Finish.
  - a. Click on Mesh
  - b. Click on Update
  - c. Close Mesher

Now, the Mesh cell in the Workbench Scheme should be checked green.

Documentation: Help -> ANSYS Workbench Help -> Meshing ->

**How to compute the case using Fluent** Start Fluent: DoubleClick on "Setup" in the Workbench Scheme

A:MyFirstTest FLUE	NT [2d, pbns, lam] [ANSYS Aca	idemic Research CFD]	
File Mesh Define Solve	Adapt Surface Display Report P	Parallel View Help	
🗃 • 🖬 • 🗟 🔘	S 💠 @ 🕀 🥕 🍭 🎗 🖪 '	• 🗖 •	
Problem Setup	Models 18	1: Mesh	
General Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Multiphase - Off Energy - Off Viscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Mehm - Off Acoustics - Off	Image: Nodel     Ic       Model     Ic       Invisoid     Image: Nodel       Spalart-Allmann (1 eqn)     0.09       K-ension (2 eqn)     C1-Epsion       K-omega (2 eqn)     1.44       Transition ST (4 eqn)     C2-Epsion       Rewnolds Stress (5 eqn)     C2-Epsion	
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Benorts	Edk	Image: Standard Original Standard Original Standard Original Standard Original Standard Original Standard Wall Functions     Image: Standard Null Functions       Image: Non-Equilibrium Wall Functions     Turbulent Viscosity Original Standard Wall Functions	Mar 18, 2010 2.1 (2d, pbns, lam)
керинз	Help 1b	CEnhanced Wall Treatment User-Defined Wall Functions TKE Prandtl Number TOR Prandtl Number Tone V	
		OK Cancel Help Writing inlet (type velocity-inlet) (mixture	. Done. (mixture) . ) Done. ~

- 1. Set Standard k-eps model
  - a. Click on Model
  - b. Click Viscous Laminar Edit
  - c. Choose k-epsilon

A:MyFirstTest FLUE File Mesh Define Solve	NT [2d, pbns, ske] [ANSYS Academic Resear Adapt Surface Display Report Parallel View H	rch CFD]
) 📸 • 🖬 • 🗟 🔘	\$ ↔ @ @ ↗ ] @ Հ 懦・□・	
Problem Setup	Boundary Conditions	1: Mesh
General Models Materials Phases Cell Zone Conditions Zoundssy Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Methods Solution Methods Solution Methods Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Zone interior-sufface_body object outlet symmetry 2a 2b 2c Phase ymmetry Edt Copy Profiles Display Mesh Periodic Conditions Help	Velocity Inlet Zone Name Inlet Zone Name Inlet Womentum Thermal Radiation Species DPM Multiphase UDS Velocity Specification Method Magnitude, Normal to Boundary Reference Frame Absolute Velocity Magnitude (m/s) 1 constant Urbulence Urbulatbod Intensity and Length Scale Urbulent Length Scale Urbulent Length Scale (m) 1 (m) (K) Cancel Help Preparing mesh for display Done. Writing Settings file Writing Settings file Writing surface_body (type fluid) (mixture) Done. Writing interior-surface_body (type interior) (mixture) (m) (K) Cancel Help (K

- 2. Inlet boundary conditions
  - a. Boundary Conditions -> Inlet: Type = velocity-inlet. Edit
  - b. Set velocity
  - c. Set turbulence intensity and length scale
- 3. Other boundary conditions
  - a. Outlet: Type=pressure-outlet, Gauge Pressure=0
  - b. Upper and Lower: Type=symmetry
  - c. Object: Type=wall

A:MyFirstTest FLUE	NT [2d, pbns, ske][ANSYS Academic Rese	arch	CFD]			
File Mesh Define Solve	Adapt Surface Display Report Parallel View	Help				
Problem Setup	Solution Initialization		1: Mesh	×		
General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Mesh Solution Controls Monitors Solution Controls Monitors Calculation Activities Run Calculation Results Graphics an Animations Plots	Compute from Inlet Reference Frame Absolute Initial Values Gauge Pressure (pascal) 0 X Velocity (m/s) 1 Y Velocity (m/s) 0 Turbulent Kinetic Energy (m2/s2) 0.015	×	4 Mesh		Mar 18 ANSYS FLUENT 12.1 (2d, pbns	, 2010 s, lam)
4a	Initialize     Reset       Reset DPM Sources     Reset Statistics	c	zon Lone. Freparin Done. Writing	nes, object symmetry outlet inlet surface_body interior-surface_body ng mesh for display Settings file writing rp variables Done.		×

- 4. Initialize solution to the value at the inlet boundary
  - a. Solution Initialization: Use "Standarad Initialization"
  - b. Compute from: inlet
  - c. Press Initialize

A:MyFirstTest FLUE	NT [2d, pbns, ske] [ANSYS Academ	ic Research CFD]	_	
File Mesh Define Solve	Adapt Surface Display Report Paralle	el View Help		
Problem Setup General Models Materials Phases Cell Zone Conditions	Residuals, Statistic and Force Monitors Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Drag - Off UR - Off	1: Mesh	Equations	
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls	Moment - Off Edit Strate Monitors	✓ Print to Console     ✓ Plot     Window     1     (	restulation         monitor Crieck Curvergence Absolute Criechter           continuity         V           x-velocity         V           y-velocity         V           k         V	
Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Create Edt Delete Volume Monitors	Iterations to Store	Residual Values Convergence Criterion Normalize Berations S V Scale Plot Renormalize Cancel Help	];
	Create Edt Delete	outlet inlet surface_bo interior-s Done. Preparing mesh for Done. Writing Settings f writing rp	dy urface_body display ile variables Done.	

- 5. Monitor convergence
  - a. Monitors -> Residuals -> Edit
  - b. Uncheck "Check convergence" for all equatios (also epsilon, you need to scroll down).

A:MyFirstTest FLUE	NT [2d, pbns, ske][ANSYS Academic Researc	ch CFD]	_ 🗆 🔀
File Mesh Define Solve	Adapt Surface Display Report Parallel View He	elp	
Problem Solve General Models Materials Phases Cell Zone Conditions Boundary Anditions Mesh Interviews Dynamic Nesh Reference Values Solution Methods Solution Methods Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Adapt Sufface Display Report Parallel View H Adapt Sufface Display Report Parallel View H Adapt Sufface Display Report Parallel View H Monitors Residuals, Statistic and Force Monitors Residuals, Statistic and Force Monitors Edit Greate Edit Delete Help	I: Mesh Options Opt	Mar 18, 2010 2d, pbns, lam) None. Done. Done. Done.
		<pre></pre>	~

- 6. Monitor drag
  - a. Monitor -> Create -> Drag
  - b. Choose object
  - c. Check "Plot"
  - d. Axes -> Y -> Uncheck Auto Range, set Range 0 to 2
  - e. Also set correct Reference Values:
    - i. "Area" is reference are for Cd in x-direction.
    - ii. For 2D it is per meter in z-direction
    - iii. Example: Height=1.0m gives Area=1.0m<sup>2</sup>
- Set first order upwind (too dissipative for giving a good solution, but might be useful for the first try since it improves the stability)
  - o Solution Methods:
  - o Spatial Discretization -> Momentum -> First Order Upwind

💶 A:MyFirstTest FLUENT [2d, pbns, ske] [ANSYS Academic Research CFD]								
File Mesh Define Solve Adapt Surface Display Report Parallel View Help								
🚔 • 📓 🔘   💲  ℚ ❶ 🖉   ℚ 久 開 • 🗆 •								
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Nethods Solution Controls Monitors Solution Controls Monitors Solution Activities Results Graphics and Animations Plots Reports	Run Calculation       Check Case       Number of Iterations       S00       1       •       Profile Update Interval       1       •       Data File Quantities       Calculate	1: Scaled Residuals	**					
	Hep	0         50         100         150         200         250         300         350         400         450           Iterations           Scaled Residuals           Mar18,20°           ANSYS FLUENT 12.1 (2d, pbms, rk           441 1.4729e-08         8.8788e-08         4.6713e-08         2.6275e-07         1.0755e-06         0:           441 1.4729e-08         8.9645e-08         4.6713e-08         2.5543e-07         1.0755e-06         0:           443 1.4937e-08         8.9694e-08         4.5713e-08         2.552e-07         1.0311e-06         0:           443 1.4937e-08         8.0904e-08         4.5719e-08         2.4554e-07         1.0154e-06         0:           444 1.4335e-08         8.6370e-08         4.4378e-08         2.3672e-07         9.468e-07         0:           445         1.4731e-08         8.6159e-08         4.3678e-08         2.3672e-07         9.4216e-07         0:           447         1.47125e-08         8.4090e-08         4.2605e-08         2.2697e-07         9.4216e-07         0:           448         1.4211e-08         8.1695e-08         4.2014e-07         9.9884e-07	(9)					

- 7. Run Calculation
  - a. Set Number of Iterations = 500 (or 1000)
  - b. Press "Calculate", Iterate until Cd levels out
  - c. Check both "Scaled Residuals" and "Drag Convergence History"

A:MyFirstTest FLUENT [2d, pbns, ske] [ANSYS Academic Research CFD]							
) 💕 • 🛃 •	• 💿 🞯 🛛	G ∲ Q	④∥ @洗腊・□・				
Problem Setup		Graphics	and Animations		1: Contours of Velocity Magn 🗸		
General Models Materials Phases Cell Zone C Boundary ( Mesh Infree Dynamic M Reference Solution M Solution Co Monitors Solution Co Monitors Solution Calculation Run Calculation Results	Ap Gra Gra Gra Conditions Conditions Conditions Pal Contours Pal Contours Options Options Options Options Global Range Clip to Range Draw Profiles Draw Mesh	Graphics Graphics Mesh Contours Vectors Pathlines Pathlines Pathlines Pathlines Pathlines Pathlines Pathlines Pathlines Pathlines S	Contours of Contours of Velocity Velocity Magnitude Min (m/s) 0 1.364986 Surfaces Enterior-surface body		1.36e+00 1.23e+00 1.09e+00 9.55e-01 8.19e-01 6.82e-01 5.46e-01 4.09e-01 2.73e-01 1.36e-01 0.00e+00 ontours of Velocity Magnilude (m/s) Mar 18, 2010		
Reports	Levels 2 20 (*) Surface Name	Setup 1 Pattern Match Display	New Surface  Surface Types axis axis clip-surf chasust-fan fan Compute Close Help		441       1.4729e-08       8.8788e-08       4.7439e-08       2.6275e-07       1.0755e-06       9:         442       1.4743e-08       8.9645e-08       4.6713e-08       2.5552e-07       1.0755e-06       9:         443       1.4937e-08       8.9645e-08       4.6713e-08       2.5552e-07       1.0851e-06       0:         443       1.4937e-08       8.9645e-08       4.5719e-08       2.5552e-07       1.0851e-06       0:         444       1.4335e-08       8.6370e-08       4.5750e-08       2.4564e-07       1.0154e-06       0:         444       1.4337e-08       8.6791e-08       4.374e-08       2.3170e-07       9.9446e-06       0:         445       1.4731e-08       8.6155e-08       4.2637e-07       9.7110e-07       0:         446       1.4731e-08       8.4900e-08       4.2605e-08       2.2607e-07       9.4216e-07       0:         447       1.4125e-08       8.2936e-08       4.1646e-08       2.2384e-07       0:       0:         nterrupting       449       1.4171e-08       8.1695e-08       4.0743e-08       2.2010e-07       8.9595e-07       0:         one.		

- 8. Plot Velocity field:
  - a. Graphics and Animations -> Contours -> Set Up
  - b. Plot Velocity Magnitude. Check "Filled"
  - c. Zoom by dragging using middle mouse. Zoom out by dragging to upper left.
  - d. Try to plot other properties (Turbulence Kinetic Energy, Velocity arrows)
- 9. Try to refine calculation:
  - a. Solution -> Solution Methods -> Momentum: Second Order Upwind
  - b. Models -> Viscous: Realizable k-epsilon

In particular, look at the amount of turbulence around the stagnation point and the differences between std k-eps and realizable k-eps.

- c. Adapt -> Gradients of Velocity
  - i. Press "Compute": Gives min and max values of velocity gradient
  - ii. Set "Refine Threshold" to something smaller than max
  - iii. Press "Mark": Gives no of cell marked for refinement
  - iv. Press "Adapt": Refines the mesh
  - v. Look at the mesh: Graphics and Animations -> Mesh -> Set Up

Documentation: Help -> User's Guide Contents (Getting Started Guide, User's Guide, et.c.)