























F2 Mesh Generation Process

Version 1

April 2025

Open The FluSol

Extreme SSD (I:) > ESDC >					Search ESDC
  Sort  View ...					
Name	Date modified	Type	Size		
 <u>GAS.VRF</u>	<u>10/12/2023 11:07 AM</u>	<u>File folder</u>			
 <u>gms</u> <u>h</u>	<u>10/12/2023 11:07 AM</u>	<u>File folder</u>			
 <u>gui</u>	<u>10/12/2023 11:07 AM</u>	<u>File folder</u>			
 <u>Manual</u>	<u>10/12/2023 11:07 AM</u>	<u>File folder</u>			
 <u>Python24</u>	<u>10/12/2023 11:07 AM</u>	<u>File folder</u>			
 <u>command_line</u>	<u>1/9/2025 11:39 AM</u>	<u>Windows Batch ...</u>	<u>1 KB</u>		
 <u>CREDITS</u>	<u>8/19/2008 12:16 AM</u>	<u>Text Document</u>	<u>6 KB</u>		
 <u>cvrt</u>	<u>10/1/2009 9:19 PM</u>	<u>Application</u>	<u>661 KB</u>		
 <u>esdc96</u>	<u>4/7/2025 12:17 PM</u>	<u>BLK File</u>	<u>1 KB</u>		
 <u>flusol</u>	<u>10/10/2023 5:10 PM</u>	<u>Application</u>	<u>3,361 KB</u>		
 <u>flusol_asy</u>	<u>3/15/2025 4:06 AM</u>	<u>Application</u>	<u>5,250 KB</u>		
 <u>hostid</u>	<u>4/9/2009 12:35 AM</u>	<u>Application</u>	<u>261 KB</u>		
 <u>LICENSE</u>	<u>8/19/2008 12:16 AM</u>	<u>Text Document</u>	<u>19 KB</u>		
 <u>Q</u>	<u>3/14/2004 7:07 PM</u>	<u>Application</u>	<u>95 KB</u>		
 <u>README</u>	<u>8/19/2008 12:17 AM</u>	<u>Text Document</u>	<u>2 KB</u>		
 <u>run-idle</u>	<u>8/25/2009 11:11 PM</u>	<u>Windows Batch ...</u>	<u>1 KB</u>		
 <u>setup</u>	<u>6/4/2016 10:28 PM</u>	<u>Windows Batch ...</u>	<u>1 KB</u>		
 <u>TUBE</u>	<u>8/19/2008 12:17 AM</u>	<u>Application</u>	<u>397 KB</u>		
 <u>tube_1</u>	<u>5/9/2009 6:15 AM</u>	<u>Windows Batch ...</u>	<u>1 KB</u>		

Double click setup to launch the flusol gui



Mesh Generation

FluSol

Solver



Post-Processing

Exit

READ	Solver	Control	Materials	I.C. & B.C.	Particles	Reactions
------	--------	---------	-----------	-------------	-----------	-----------

Read mesh and CFD model files

☐ 6. Open *.da ☐ 7. New model file name

☒ 3. Open *.msh ☒ 4. New model file name

☐ 1. Open *.cbk

Convert Model

☐ Nastran model *.nas ----> FluSol model *.da

☐ HyperMesh *.hmaskii ----> FluSol model *.da

Export Model

☒ FluSol model *.da ----> gmsh geometry *.geo

☐ FluSol model & result (pl.res) ----> gmsh result *.pos

☐ FluSol model *.da ----> HyperMesh mesh *.hmaskii

1. Click 3 to enter mesh file
I:\ESDC\GAS.VRF\3D\f2-blunt-nose-cylinder\2025-04-06\F22d.msh
2. Click 4 to provide a filename for storing
2d mesh file such as 2d.da
3. Click 5_1 to generate 2d.da model file

Read mesh and CFD model files

(1) 6. Open *.da (2) 7. New model file name (5) 8. Run FluSol

I:\ESDC\GAS.VRF\3 3d.da

3. Open *.msh 4. New model file name

1. Open *.cbk 2. Create *.msh file

View Mesh

Convert Model

Nastran model *.nas ----> FluSol model *.da

HyperMesh *.hmascii ----> FluSol model *.da

Export Model

FluSol model *.da ----> gmsh geometry *.geo

FluSol model & result (pl.res) ----> gmsh result *.pos

FluSol model *.da ----> HyperMesh mesh *.hmascii

Write model

Preview model

EXIT

5_1. Create 2D and 3D Model without wings

5_2. Create (a) ax.da and (b) Wing-Body combination 3d file

Solver Block

☒ Title description

mach 1.6 flow past blunt-nose-cylinder at zero angle of attack

Solver Compressible Flow

Dimension Axisymmetric

Arti pressure (4)

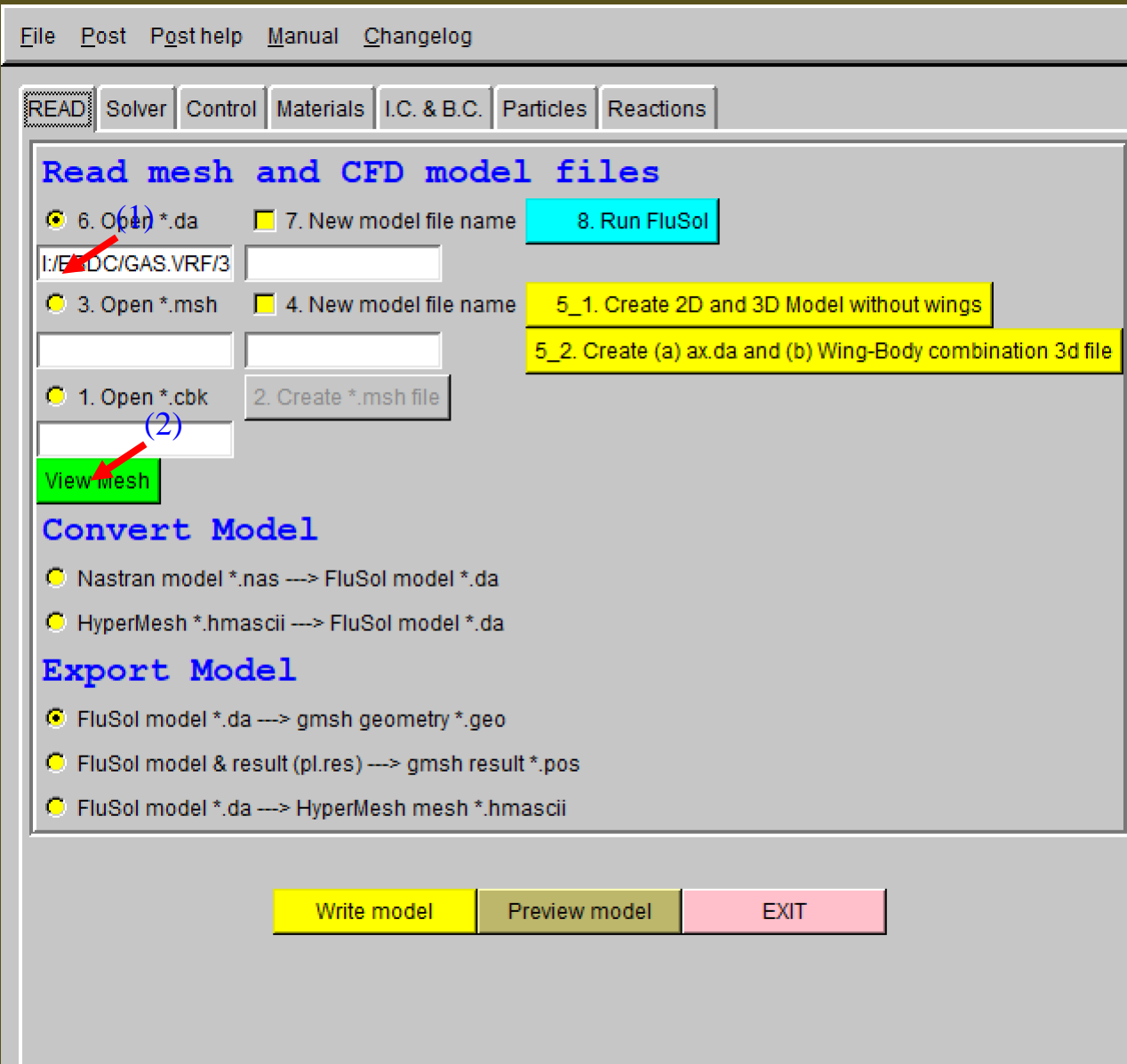
Memory store in ram

Convert sweep axis line

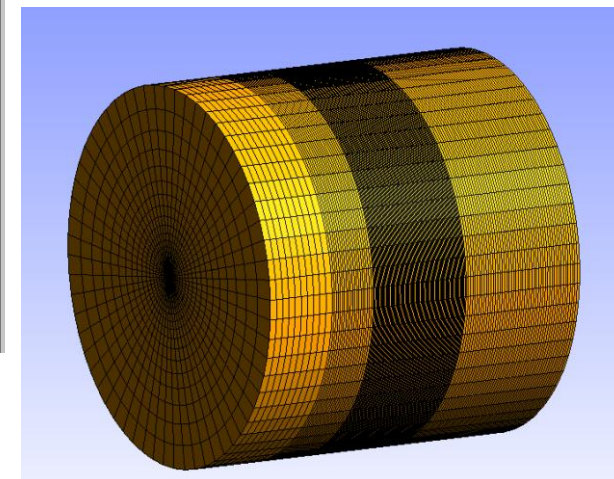
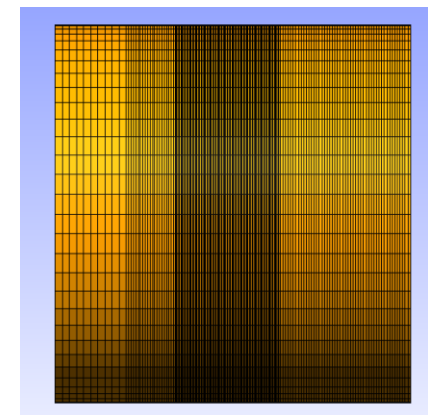
File bln

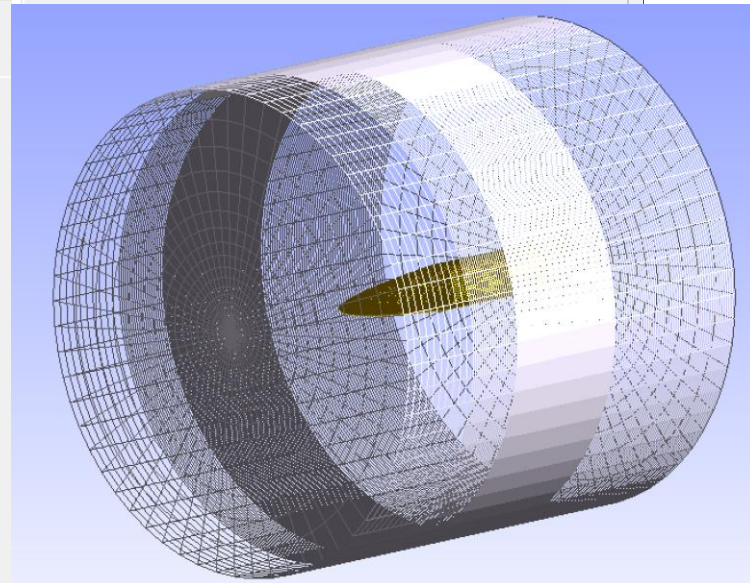
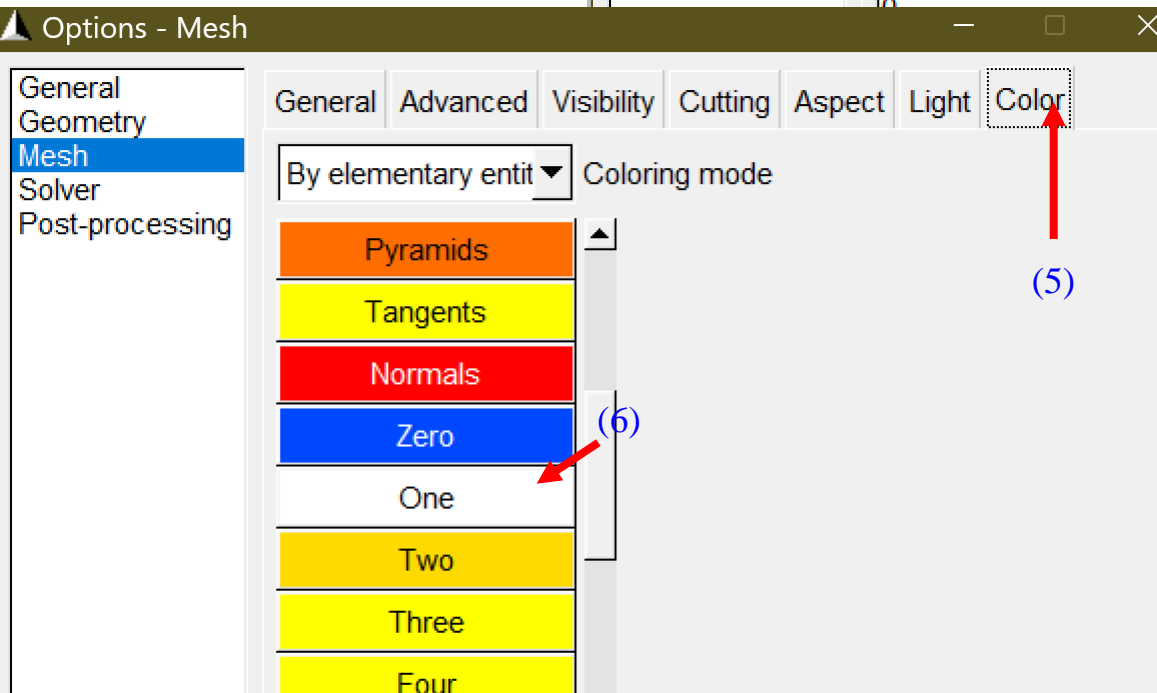
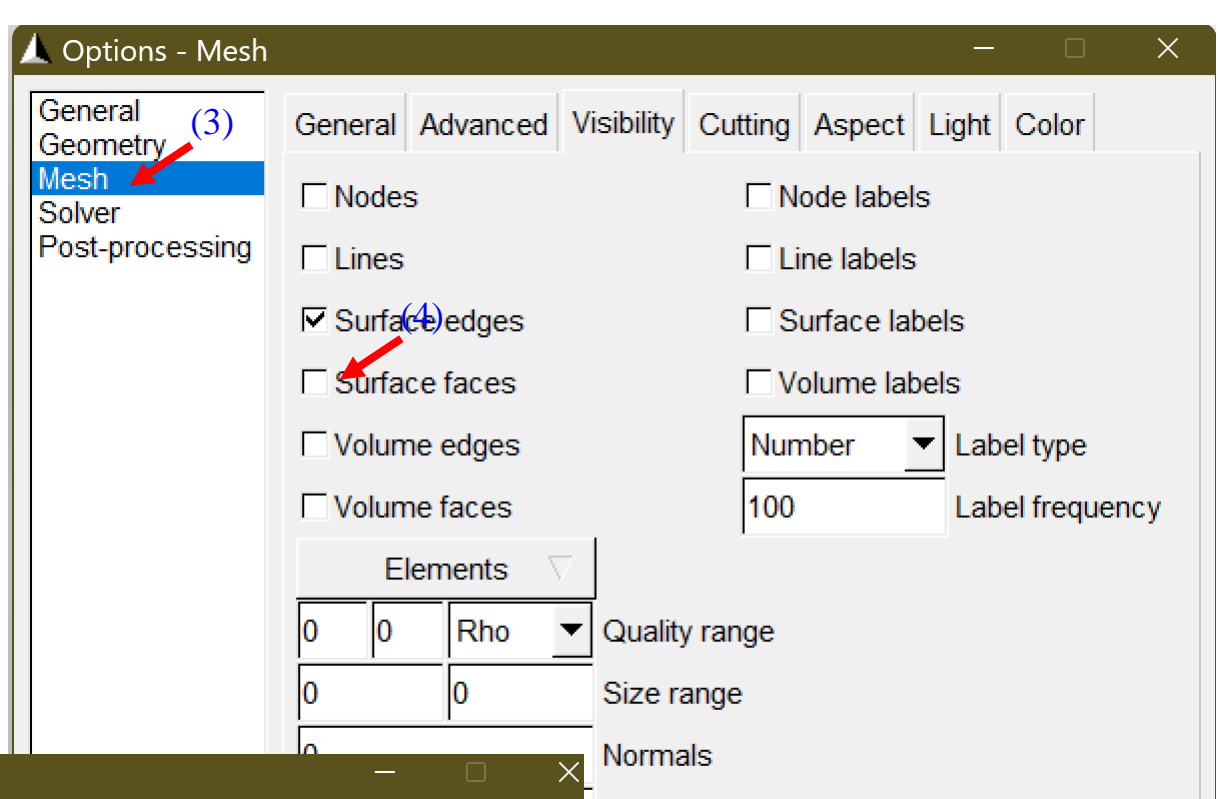
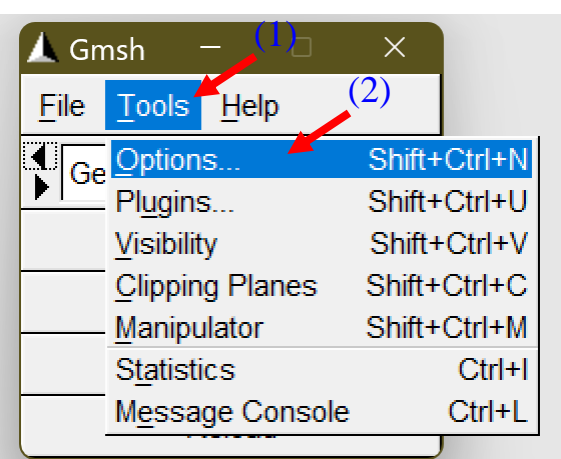
1. Click 6 to enter 2d.da model file I:\ESDC\GAS.VRF\3D\2-blunt-nose-cylinder\2025-04-06\2d.da
2. Click 7 to provide a filename for storing 3d model file such as 3d.da
3. Click solver button
4. Click convrt to select "sweep axis line"
5. Click 8 run solver to generate 3d.da model file

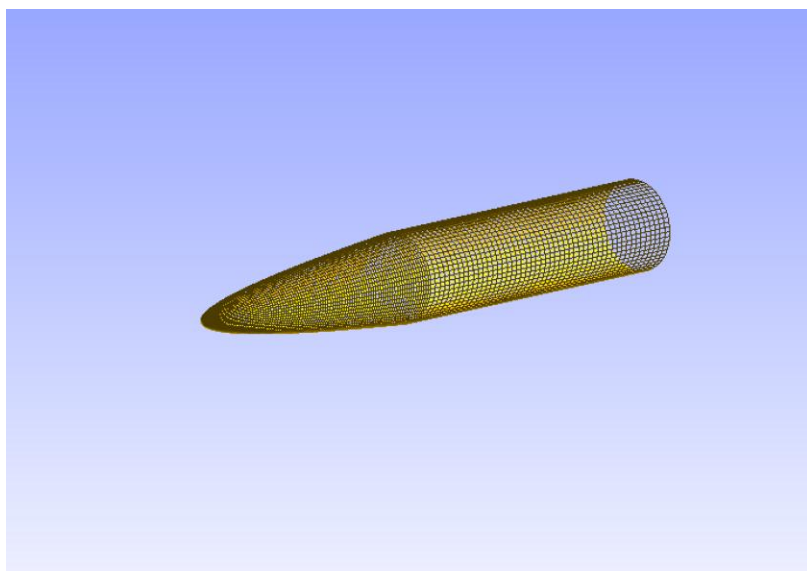
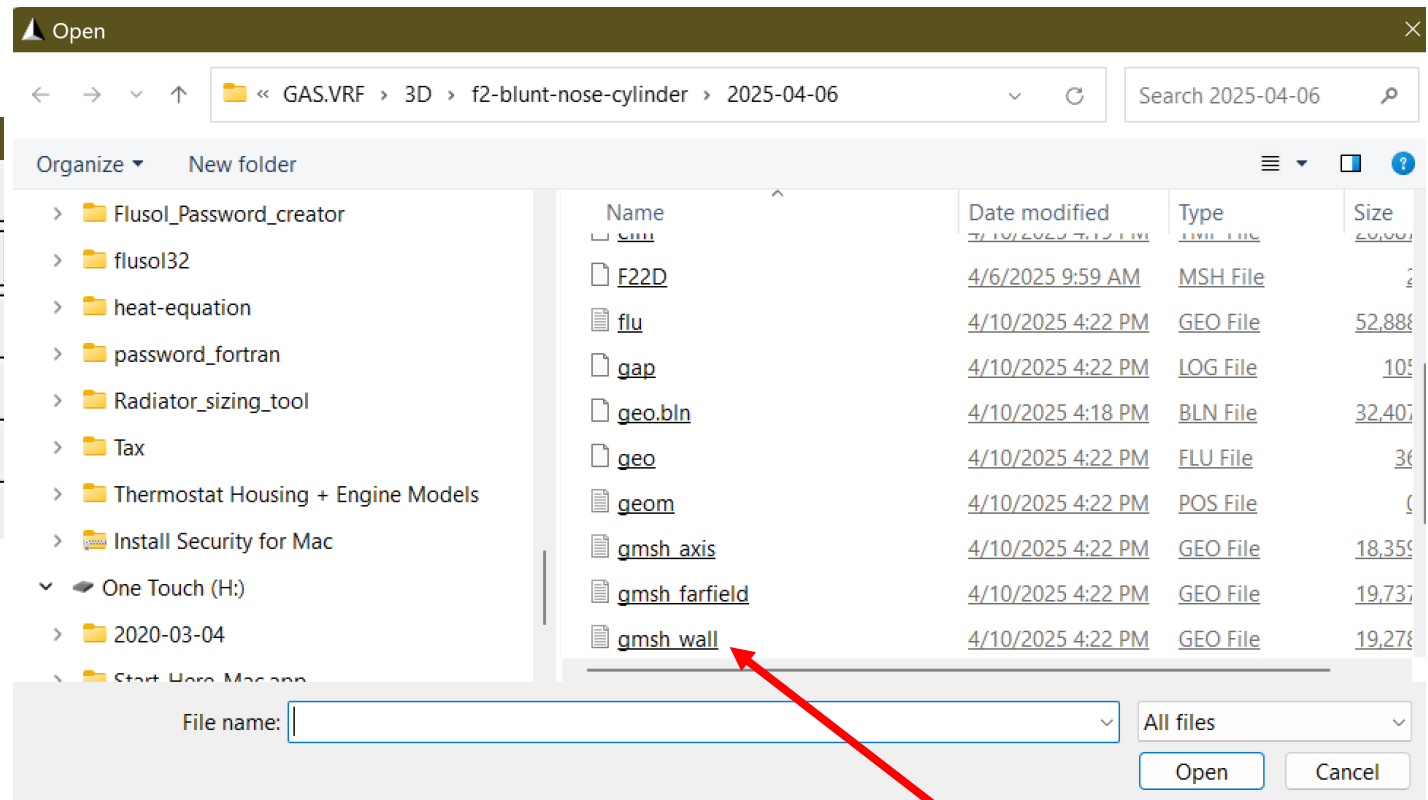
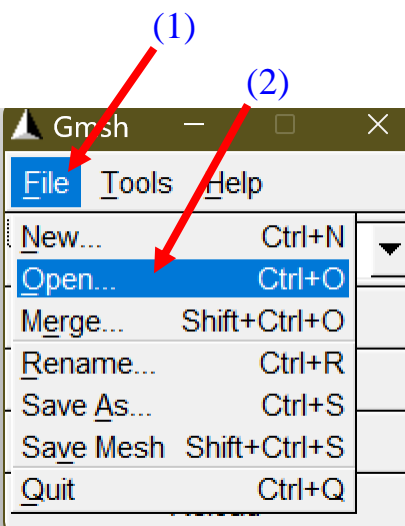
(3)



1. Click 6 to enter 3d.da model file I:\ESDC\GAS.VRF\3D\f2-blunt-nose-cylinder\2025-04-06\3d.da
2. Click "View Mesh" to plot the mesh
3. Click solver button
4. Click convrt to select "sweep axis line"
5. Click 8 run solver to generate 3d.da model file







☒ READ
 ☒ Solver (1)
 ☐ Control
 ☐ Materials
 ☐ I.C. & B.C. (2)
 ☐ Particles
 ☐ Reactions

Read mesh and CFD model files (4)

☒ 6. Open *.da
 ☐ 7. New model file name
 ☒ 8. Run FluSol

I:/ESDC/GAS.VRF/3

☐ 3. Open *.msh
 ☐ 4. New model file name
 ☐ 5_1. Create 2D and
 ☐ 5_2. Create (a) ax.da

☒ 1. Open *.cbk
 ☐ 2. Create *.msh file

View Mesh

Convert Model

☐ Nastran model *.nas ----> FluSol model *.da
☐ HyperMesh *.hmascii ----> FluSol model *.da

Export Model

☒ FluSol model *.da ----> gmsh geometry *.geo
☐ FluSol model & result (pl.res) ----> gmsh result *.pos
☐ FluSol model *.da ----> HyperMesh mesh *.hmascii

Initial and Boundary Conditions

<input checked="" type="checkbox"/> *element type	View
<input type="checkbox"/> *mesh	View
<input type="checkbox"/> *sweep	View
<input type="checkbox"/> *chemical	View
<input type="checkbox"/> *plane	View
<input type="checkbox"/> *bvlr	View
<input checked="" type="checkbox"/> *initial conditions (3)	View
<input checked="" type="checkbox"/> *face of boundary conditions	View
<input type="checkbox"/> *boundary conditions conditions	View
<input checked="" type="checkbox"/> *output control cards	View
<input type="checkbox"/> *ckinitial conditions	View
<input type="checkbox"/> *ckbcd conditions	View
<input type="checkbox"/> *particle initial conditions	View
<input type="checkbox"/> *trace particles	View
<input type="checkbox"/> *material property	View

Complete numerical integration

Space for boundary conditions = 0 words
Space for shape functions = 6718140 words
Space for input data = 1608827 words
Space for result data = 0 words
End of store

Total disk space needed for this run = 8326967 Words
Total disk space needed for this run = 31.765 Mega-Byte
Total CPU (RAM) used for this run = 46683605 Words
Total CPU (RAM) used for this run = 178.084 Mega-Byte

Total time for this run = 1796 seconds

----- FluSol -----

----- A General Purpose Fluid Flow Solver -----

----- 3D Computational Fluid Dynamics Program -----

----- Version 3, Jan. 2008 -----

----- (c) Copyright 1996 -----

FluSol CFD Version 3.0, April 2007



Mesh Generation

FluSol

Solver



Post-Processing

