# ADVENTURE\_Magnetic\_on\_Windows

Version 0.3b User's Manual

2019/09/17

# **ADVENTURE** Project

Copyright © ADVENTURE Project 2016-2019 All rights reserved

# **Table of Contents**

1	Ove	erview	.3
	1.1	Introduction	. 3
	1.2	About ADVENTURE_Magnetic_on_Windows	. 3
	13	Structure of this Manual	3
	1.0	Multithreading of Colver	. U
_	1.4		.o _
2	Noi	n-linear Magnetostatic Analysis	. 5
	2.1	Model	.5
	2.2	Divide Coil Domain	.6
	2.3	Starting the Program	.7
	2.4	Creating Analysis Case	.9
	2.5	[Preprocess 1] Mesh Generation1	2
	2.5.	1 Selection of CAD Models 1	12
	2.5.	2 Settings for Node Density 1	<b>4</b>
	2.5.	3 Create Surface Patch 1	15
	2.5.	4 Mesh Generation	15
	2.6	[Preprocess 2] Setting Analysis Conditions 1	18
	2.6.	1 Setting Material Property Values 1	18
	2.6.	2 Setting Boundary Condition	22
	2.6.	3 Solver input file creation	25
	2.7	Analysis Execution	26
	2.7.	1 Domain Decomposition	26
	2.7.	2 Solver Execution	27
	2.8	Result Visualizing	32
	2.8.	1 Start Visualizing	32
	2.8.	2 Terminating AdvMagOnWin	32
	2.8.	3 Launching ParaView and Reading Files	33
	2.8.	4 ParaView 3D screen operation method	34
	2.8.	5 Mesh Display	35
	2.8.	6 Display of Magnetic Flux Density	35
	2.8.	7 Display of Electromagnetic Force	38
3	Tin	ne-harmonic Eddy Current Analysis	<i>39</i>
	3.1	About the model used	39
	3.2	Starting the Software4	<b>!1</b>
	3.3	Creating Analysis Case	11

<b>3.4</b>	[Preprocess 1] Mesh Generation	
3.4.1	1 Selection of CAD Models	
3.4.2	2 Settings for Node Density	
3.4.3	3 Create Surface Patch	
3.4.4	4 Mesh Generation	
3.5	[Preprocess 2] Setting Analysis Conditions	47
3.5.1	1 Setting Material Property Values	
3.5.2	2 Setting Boundary Condition	53
3.5.3	3 Solver input file creation	
3.6	Analysis Execution	57
3.6.1	1 Domain Decomposition	
3.6.2	2 Solver Execution	59
3.7	Result Visualization	62
3.7.1	1 Start of Visualization	
3.7.2	2 Terminating AdvMagOnWin	
3.7.3	3 Launching ParaView and Reading Files	
3.7.4	4 Method of 3D Screen Operation of ParaView	
3.7.5	5 Displaying Mesh	
3.7.6	6 Displaying Magnetic Flux Density	
4 Use	ful Functions	
4.1	Mesh Extraction of Selected Volume	72
4.2	Creation of B-H curve definition file	
4.2.1	1 Startup	
4.2.2	2 Adding definition values	
4.2.3	3 Delete definition value	
4.2.4	4 Reading the definition file	
4.2.5	5 Saving the B-H curve definition file	
4.2.6	6 Creation complete	

# 1 Overview

### 1.1 Introduction

This manual explains the steps of electromagnetic field analysis using ADVENTURE\_Magnetic\_on\_Windows.

# 1.2 About ADVENTURE\_Magnetic\_on\_Windows

ADVENTURE\_Magnetic\_on\_Windows ("AdvMagOnWin") is a software for performing electromagnetic field analysis on Windows using ADVENTURE\_Magnetic developed in the ADVENTURE project. AdvMagOnWin provides functions to import CAD files, generate mesh, set material properties, set boundary conditions, decompose domain, set analysis conditions, execute solvers, and create visualization files using GUI based on the user interface agent software "ADVENTURE\_iAgent".

Among the analysis functions supported by ADVENTURE\_Magnetic Ver. 1.7.0, AdvMagOnWin provides non-linear magnetostatic field analysis and time-harmonic eddy current analysis.

# 1.3 Structure of this Manual

This manual has the following structure.

Chapter 1 Overview

In this chapter, the overview of AdvMagOnWin is explained.

Chapter 2 Non-linear Magnetostatic Analysis

In this chapter, the steps of non-linear magnetostatic analysis of AdvMagOnWin is explained by using the attached sample data.

Chapter 3 Time-harmonic Eddy Current Analysis

The procedure of the time-harmonic eddy current analysis by AdvMagOnWin is explained using the attached sample data.

Chapter 4 Useful functions

In this chapter, utilities provided by AdvMagOnWin. are described.

#### 1.4 Multithreading of Solver

AdvMagOnWin is equipped with OpenMP multi-thread solver.

At startup, AdvMagOnWin automatically sets the number of logical processors recognized by Windows as the number of threads used by the solver. The number of logical processors can be checked on the "Performance" tab of "Task Manager" (Fig. 1.4-1).

If you want to specify an arbitrary number of threads, edit advmagonwin.bat as follows and start AdvMagOnWin.

Before change:

```
set OMP_NUM_THREADS =% NUMBER_OF_PROCESSORS%
```

After change (example when the number of threads is 4): set OMP\_NUM\_THREADS = 4



Fig. 1.4-1 Task Manager

# 2 Non-linear Magnetostatic Analysis

In this section, we describe the steps of non-linear magnetostatic analysis using sample data attached to AdvMagOnWin.

## 2.1 Model

The model used in this chapter is a suction magnet that has a space where iron cores with different top and bottom shapes face each other. Fig. 2.1-1 shows the cross section of this model. The unit used in Fig. 2.1-1 is [mm]. The analysis area is the area obtained by rotating Fig. 2.1-1 around the z axis by 10 degrees. Suppose that the forced current density of the coil part flows in the direction perpendicular to the cross section in Fig. 2.1-1 at  $3 \times 10^7$  [A/m<sup>2</sup>]. Magnetic reluctivity of air and coil  $\nu$  is  $1/(4\pi) \times 10^7$  [m/H]. The material of annular magnet is SS41P, and in this area it shall have non-linearity as shown in Fig. 2.1-2. A list of property values is shown in Table 2.1-1. The boundary condition imposes A × n = 0 (where A is the magnetic vector potential [Wb / m], n is the unit normal vector of the boundary surface) on all surfaces.





Fig. 2.1-2 B-H Curve

Table 2.1-1 List of Materia	al Properties
-----------------------------	---------------

	<b>A</b>	
Material	Magnetic Reluctivity	B-H Characteristic
Coil	795774.7	-
Magnetic Material	757.1	-
air	795774.7	Apply Fig. 2.1-2's characteristic curve

# 2.2 Divide Coil Domain

Since most of the models used in magnetic field analysis consist of multiple materials, AdvMagOnWin uses ADVENTURE\_TriPatch's surface patch merging program mrpach to combine multiple surface patches internally. However, if the shape of the bonded surface between the materials is mismatched, the join may fail. In such a case, it may be successful to divide the material into two and then create an IGES file to match the shape of the join plane.

In the model used this time, the merging surfaces of the materials do not match in the right part of the coil (x = 85 mm in Fig. 2.2-1). Therefore, you will divide the coil as shown in Fig. 2.2-1, and make the merging surfaces coincide.



Fig. 2.2-1 Decomposition of Coil Domain

## **ADVENTURE SYSTEM**

#### 2.3 Starting the Program

Double click "advmagonwin\_en.bat" in AdvMagOnWin installation folder and start AdvMagOnWin. When you start up, a window shown in Fig. 2.3-1 will appear.

MADVENTURE_Magnetic_on_Windows				
	art :_on_Windo			
ADVENTURE				
	Start			

Fig. 2.3-1 Start Window

Note that if you run advmagonwin\_en.bat on Windows 8 / 8.1 / 10, it may show the screen shown in Fig. 2.3-2. This is a screen to check if there is any danger to the application you tried to start up, and it will be shown when you run for the first time. Clicking the "More info" in Fig. 2.3-2 will change to the screen shown in Fig. 2.3-3. You can activate AdvMagOnWin by clicking "Run anyway" in Fig. 2.3-3.



Fig. 2.3-2 The Screen Shown When Executing the Batch File



Fig. 2.3-3 The Screen Shown after Clicking "More info"

Click the "Start" button to show windows in Fig. 2.3-4.

M ADVENTURE_Magnetic_on_Windows - X	
Elle Mesh Analysis Help	
Current analysis type :	
Analysis Type: Magnetic Field Analysis - F	
Cluster Usage Single Processor	
File Name: (To be saved)	
A) Monu Window	
A) Wenu w muow	
To create new analysis	
case, dick [File] - [Create	
New Analysis Case,	
Cick File) - Load Analysis	
AQUENTURE Case]	
To select ICES file, click	
C C preserve coronary	
D) Massas Window	
B) Message window	
W Operatio X	
start	
Read IGES file	
Set node density	
Make surface patch	
Make mesh	
Haladal Departs Sellings	
set boundary condition	
Make Adventure input the	
Domain decompose C) Operation Flow Window	
Run slover	
Export Analysis Result	
Display seal	
Advintagun Willin	
🟥 🖸 ここに入力して検索 🛛 🔱 💷 🤮 🧱 🔂 💷 📓 📾	~ 12 40) A 2017/10/17 ↓

Fig. 2.3-4 Initial Windows of AdvMagOnWin

A) Menu Window

This is a window to call all operations related to analysis. At the bottom of the window, the current analysis type is shown.

- B) Message Window
   Advice from the agent (concrete operation method and information on the current operation) is shown.
- C) Operation Flow Window

Current operation plan is shown. If you click the button next to each item, a summary of the operation with that item will be shown in the message window.

# 2.4 Creating Analysis Case

In order to perform a new analysis, you will create an "analysis case". To create analysis case, select "<u>File</u>" -> "Create <u>New Analysis Case</u>" in the menu window.

First of all, you are asked "Save current Analysis?" (Fig. 2.4-1). If you are currently opening the analysis case, select "Yes", but here you select "No" because you just started the program.

Save cu	×				
Save current Analysis?					
Yes <u>N</u> o Cancel					

Fig. 2.4-1 Confirmation of Saving Analysis Case

Next, the window in Fig. 2.4-2 appears. Please click "Next".

🗯 Start Analysis Planning		_	C	]	×
Start About Analysis Type About Analysis Subtype About Element Type E End	In following windows, you enter analysis ty element type. Then I will make analysis pla Start by clicking "Next" button.	be, usage of o	cluste	r and	
	Canc	el < Ba	ck	Nex	(t >

Fig. 2.4-2 Creating Analysis Case

Next, the window in Fig. 2.4-3 appears. After confirming that "Electromagnetic Field Analysis" is selected as the type of analysis case click "Next".

📾 Analysis Type			_		×
S Start About Analysis Type About Analysis Subtype About Element Type E End	Select Analysis Type.				
	·	Cancel	< Back	(	Next >

Fig. 2.4-3 Analysis Type

Next, the window in Fig. 2.4-4 appears. After confirming that "Non-linear Magnetostatic" is selected as a more detailed genre of the analysis case, please click "Next".

🕷 Analysis Type(Subtype) - Electromagnetic Field Analysis 🛛 — 🗆 🗙						
S Start About Analysis Type About Analysis Subtype About Element Type E End	Select Analysis Subtype for: Electromagnetic Field Analysis Non-linear Magnetostatic Time-harmonic Eddy Current					
		Cancel	< Back		Next >	

Fig. 2.4-4 Analysis Type (Subtype)

Next, the window in Fig. 2.4-5 appears. Select "IGES" as the geometry model, "Linear Tetrahedron (Edge Element)" as the analysis model, and click "Next".

🗯 Element Type			_		×
S Start	Select Model Type.				
About Analysis Subtype	Geometry model IGE S	Analysis model Einear Tetrahedron	(Edge Eler	nent)	
About Element Type	🔾 (Mesh)	O Read mesh file			
E End	<ul> <li>Patch</li> <li>ADV_Cad</li> </ul>	○ Tetrahedron			
		Cancel	< Back	N	lext >

Fig. 2.4-5 Element Type

An analysis case is created by these operations, and a list of necessary operations is shown in the operation flow. The subsequent operations are performed according to the operation flow and the message window. Next, we will generate a mesh.

# 2.5 [Preprocess 1] Mesh Generation

# 2.5.1 Selection of CAD Models

First, specify CAD model files as the analysis shape. If you select "Mesh" -> "Select IGES Files" in the menu window, a window like the one in Fig. 2.5-1 appears.

📾 IGES List	_	
Set IGES files.		
		add
		remove
		up
		down
	OK	Cancel

Fig. 2.5-1 IGES List Window

When clicking the "add" button in Fig. 2.5-1, the file reading dialog (Fig. 2.5-2) is shown. Select "coil01.igs" in the "<AdvMagOnWin installation folder> \ sample\_data \ shaft" folder and click "Open".

🖃 Open		×
Look <u>I</u> n:	shaft	• 6 6 6 8 5
<ul> <li>test_shaft</li> <li>air.igs</li> <li>coil01.igs</li> <li>coil02.igs</li> <li>mag.igs</li> </ul>	12.files	
File <u>N</u> ame:	coil01.igs	
Files of <u>T</u> ype:	IGES file(*.igs)	
		Open Cancel

Fig. 2.5-2 File Selection Dialog

If coil01.igs file is selected, it is added to the IGES list (Fig. 2.5-3).

🐲 IGES List	_		$\times$
Set IGES files.			
agOnWin-0.2b\sample	_data\shaft\coil0	)1.igs	add remove
			up down
•		) DK	Cancel

Fig. 2.5-3 IGES List after Selecting coil01.igs

Please add "coil02.igs", "mag.igs", "air.igs" in the similar process. The unit used in these files is [m]. If you add it all, it will look like Fig. 2.5-4. Click "OK" to load the IGES files.

🐖 IGES List	—		$\times$
Set IGES files.			
agOnWin-0.2b\sample_data agOnWin-0.2b\sample_data	i\shaft\coil i\shaft\coil	01.igs 02.igs	add
agOnWin-0.2b\sample_data	ı\shaft\mag	g.igs	remove
agOnWin-0.2b\sample_data	ı\shaft∖air.i	gs	
			up
			down
•		•	
		ок	Cancel

Fig. 2.5-4 IGES List after Selecting All Files

#### 2.5.2 Settings for Node Density

Next, specify the node density. Select "Mesh" -> "Set Node Density" (Fig. 2.5-5).

💐 Settings for	r Node Density	_		$\times$
Set node density.				
Base distance				
				1.0
Туре	Range	Intensity of Density		read file
				on Point
				on Line
				on Cylinder
				remove
		0	К	Cancel

Fig. 2.5-5 Node Density Setting Dialog

There are three types of node density designation: point designation, line segment designation, and cylinder designation. It is also possible to read an ADVENTURE standard node density setting file. The unit used in a file is [m].

In this example, load the node density setting file from sample data. Please click "read file". When the file reading dialog (Fig. 2.5-6) appears, select "shaft.ptn" in the "<AdvMagOnWin installation folder> \sample\_data\shaft" folder and click "Open".

🐲 Open		×
Look <u>i</u> n:	shaft	▼ a a a b b b b
📑 test_shaft	I2.files	
File <u>N</u> ame:	shaft.ptn	
Files of <u>Type</u> :	ptn File(*.ptn)	•
		Open Cancel

Fig. 2.5-6 Selection of Node Density Setting File

As a result of loading the contents of the file, the basic node distance is changed to 0.01 [m], and a local node density setting of the cylinder type is added (Fig. 2.5-7). Click "OK" to complete the setting.

💐 Settings for	_	C	x c	
Set node density.				
Base distance				
				0.01
Туре	Range	Intensity of Density	/	read file
Cylinder	0.11,0.1,0.07,0.0	3.0		
				on Point
				on Line
				on Cylinder
				remove
		C	K	Cancel

Fig. 2.5-7 Settings for Node Density Dialog (After File Selection)

# 2.5.3 Create Surface Patch

Next, make a surface patch from the CAD shape. When you select "<u>M</u>esh" -> "Make <u>P</u>atch", the patch creation window (Fig. 2.5-8) appears. Click "OK" to make patch

🏁 Make patch	×
Start make surface patches and merge.	
OK Canc	el

Fig. 2.5-8 Make Patch Dialog

#### 2.5.4 Mesh Generation

Next, we will generate a mesh. If you select "<u>M</u>esh" -> "<u>M</u>ake Mesh", the mesh generation window (Fig. 2.5-9) appears. If you keep checking "correct surface patch" in the window, the surface patch is automatically corrected before generating mesh. Click "OK" to start mesh generation.

🦛 Mesh Generation				
Start mesh generation.				
✓ correct surface patch				
OK Can	cel			

Fig. 2.5-9 Mesh Generation Dialog

When the mesh generation is completed, the total number of elements and nodes is shown (Fig. 2.5-10). After confirming the number of elements and nodes, please click "OK".



Fig. 2.5-10 Total Number of Elements and Nodes in Generated Mesh File

Next, we set the property value and set the boundary conditions.

2.6 [Preprocess 2] Setting Analysis Conditions

# 2.6.1 Setting Material Property Values

Set material property values from "<u>A</u>nalysis" -> "Set <u>M</u>aterial Property" -> "Electro<u>m</u>agnetic Field Analysis" (Fig. 2.6-1).

谢 Property	Data Settings		×
Set material p volume.	roperty data. When clicking a cell, a dia	log is displayed to set the physical p	roperty value of the corresponding
Property ID	Magnetic Reluctivity	Material Property Type	Applied Volume
0	795774.7 O	thers	0-3
		Add Material Properties	Remove Material Properties
- Angular Fro	Webcy		
Angular from	uoncy [rad/e] of alternating current in	coil	276.0011
miguidi ifeq	dency frames or allernating current in	coll	
			OK Cancel

Fig. 2.6-1 Property Data Settings Dialog (Initial)

In the initial screen, material property is automatically set for Property ID = 0, Magnetic Reluctivity = 795774.7, Material Property Type = Other, Applied Volume = 0-3. Edit material properties as shown in Table 2.6-1.

	Table 2.0-1 Waterial Toperty Settings and Volume Correspondence				
Property ID	Magnetic Reluctivity	Material Property Type	Applied Volumes		
0	795774.7	Coil	0-1 (coil01.igs, coil02.igs)		
1	757.1	Magnetic Material	2 (mag.igs)		
2	795774.7	Others	3 (air.igs)		

Table 2.6-1 Material Property Settings and Volume Correspondence

## **ADVENTURE SYSTEM**

First, change the Property ID 0 to the material property value of the coil.

When you click the line of the Property ID 0, material property change dialog (Fig. 2.6-2) is shown.

🗯 Material Property Cha	ange			>
Material Property Ty	Material Property Value Settings	;		
🔾 Coil	Magnetic Reluctivity			795774.7
Magnetic Material	Coil			
Permanent Magnet	Definition Type		RF	<b>•</b>
O Conductor	boundon type			
Others	Definition File			
	Magnotic Matorial			
	Definition File			
	Definition File			
	Permanent Magnet			
	Definition Type	RF		-
	Definition file 4			
	Definition File 1			
	Definition File 2			
	0			
	Conductor			
	Conductivity			
plume to be Applied 0-3				
onfigurable Volume:0-3				
Applicable volumes can 4)	be specified in both range specifica	ation and enumera	ation. Combined use	e is also possible (eg: 1,
				OK Cancel

Fig. 2.6-2 Material Property Change Dialog (Initial)

Follow the steps below to make changes.

- "Material Property Type"
   Change "Material Property Type" on the left side from "Others" to "Coil". The "coil" setting field becomes effective.
- "Coil" -> "Definition Type" Change from "RF" to "MD".
- "Coil" -> "Definition File"
   Select "coil.dat" in "<AdvMagOnWin installation folder>¥>\_\sample\_data¥\shaft" folder as "Definition File".
- 4) "Volume to be Applied"Change "Volume to be Applied" on the lower side to 0 1.

When setting changes are completed, please click "OK". The material property list is shown again (Fig. 2.6-3)

Property	Data Settings		×
Set material ; volume.	property data. When clicking a cell, a dialo	og is displayed to set the physical p	roperty value of the corresponding
Property ID	Magnetic Reluctivity	Material Property Type	Applied Volume
0	795774.7 Co	il D	)-1
		Add Material Properties	Remove Material Properties
- Angular Fre	quency		
Angular freq	uency [rad/s] of alternating current in c	oil	
L			
			OK Cancel

Fig. 2.6-3 Material Property List (After Coil Settings)

Next, set the material property value of magnetic material.

When you click "Add Material Properties", the material property setting dialog will be shown. Please select "Magnetic Material" as "Material Property Type". Next, input "757.1" to "Magnetic Reluctivity" and select "bh\_curve" in "<AdvMagOnWin installation folder>¥>\\_sample\_data¥\shaft" as "Definition File" of "Magnetic Material".

Finally, enter "2" as "Volume to be Applied".

When all settings are completed, it becomes as shown in Fig. 2.6-4. Click "OK" to return to the material property list.

## **ADVENTURE SYSTEM**

Addition of Material	Properties					×
Material Property Ty	Material Prope	erty Val	lue Settings		7	57.4
	Magnetic Relut	uvity				57.1
Magnetic Material	Coil					
<ul> <li>Permanent Magnet</li> </ul>	Definition Type				RF	-
<ul> <li>Conductor</li> </ul>						
<ul> <li>Others</li> </ul>	Definition File					
	Magnetic Mate	erial				_
	Definition File	Nhome	e\miyoshi\workspace\A	dvMag	iOnWin\sample_data\shaft\bh_curve	
		Import	t the file after completi	ng ma	terial property settings.	
	Permanent Ma	ngnet –				
	Definition Type			RF		-
						_
	Definition File 1					
	Definition File 2					
	Conductor					
	Conductivity					
Volume to be Applied 2						
Configurable Volume:2-3						
* Applicable volumes can 2-4)	be specified in b	oth ran	nge specification and er	numer	ation. Combined use is also possible (e	g: 1,
					OK Can	icel

Fig. 2.6-4 Addition of Material Properties Dialog (After Setting of Magnetic Material)

Finally, you will do the settings of the air. As in the case of the magnetic material, click on "Add Properties" and open the material property setting dialog.

Please select "Others" as "Material Property Type". Please enter "795774.7" as "Magnetic Reluctivity" and "3" as "Volume to be Applied".

After all settings are made, it will be as shown in Fig. 2.6-5. Click "OK" to return to the property data settings dialog (Fig. 2.6-6).

## **ADVENTURE SYSTEM**

Addition of Material	Properties	×
Material Property Ty	Material Property Value Settings	
🔾 Coil	Magnetic Reluctivity	795774.7
<ul> <li>Magnetic Material</li> <li>Permanent Magnet</li> </ul>	Coil	RF
Conductor	Sounder Type	
Others	Definition File	
	Magnetic Material	
	Definition File	
	Permanent Magnet	
	Definition Type	RF
	Definition File 1	
	Definition File 2	
	Conductor	
	Conductivity	
olume to be Applied 3		
onfigurable Volume:3		
Applicable volumes can 4)	be specified in both range specifica	tion and enumeration. Combined use is also possible (eg: 1,

# Fig. 2.6-5 Addition of Material Properties Dialog (After Setting of Air Area)

🥮 Prope	rty Data Settings		×
Set material p volume.	property data. When clicking a cell, a c	dialog is displayed to set the physical p	property value of the corresponding
Property ID	Magnetic Reluctivity	Material Property Type	Applied Volume
0	795774.7	Coil	0-1
1	757.1	Magnetic Material	2
2	795774.7	Others	3
		Add Material Properties	Remove Material Properties
Angular Fre	quency		
Angular freq	juency [rad/s] of alternating current	in coil	

Fig. 2.6-6 Property Data Settings (After Setting is Complete)

Please check whether there is any problem in setting and click "OK".

# 2.6.2 Setting Boundary Condition

Next, set the boundary condition. First, if you select "Analysis" -> "Set Boundary Condition", the dialog in Fig. 2.6-7 appears.



Fig. 2.6-7 Set Boundary Condition Dialog

In this case, in order to reconstruct CAD surface information lost by mesh generation, surface triangle (patch) is grouped according to its normal direction, and a surface (patch group) is extracted. Pasting boundary conditions is done in patch group units.

It is necessary to specify the two-sided included angle corresponding to the grouping resolution. The smaller the angle (the larger the value of (a) in Fig. 2.6-7), the more detailed the classification of the surface can be done. In the model of this time, there is no problem as it remains as standard, so please press "OK" as it is. The group division is performed automatically, and the window for setting the boundary condition is activated (Fig. 2.6-8).



Fig. 2.6-8 Boundary Condition Setting Window (Initial State)

At this time, the operation flow automatically changes from the list of the whole analysis steps so far to the operation steps of the boundary condition (Fig. 2.6-8 red arrow). In addition, guide messages corresponding to the operation steps are also shown in the message window, so please proceed with the operation while referring to here as well.

On the boundary condition setting window, operate the view of the model as follows.

- Rotation:
  - Hold down the wheel button (middle button) and move the mouse to rotate the model.
  - Translation: Hold the left mouse button and move the mouse to move the model in parallel.
- · Zoom:
  - Hold down the right button and move the mouse upward to zoom out or move it down to zoom in.
- Node selection:
  - Click the target node with the mouse to select the node. Yellow square is shown at the chosen node (although it is hard to see).
- Surface selection:
  - Right-click with the node selected, you can select the surface to which the node belongs. If you continue right clicking, another surface (which that node belongs to also) is selected.

After selecting the node / surface for which you want to add the boundary condition, select "BC" -> "BC (Magnetic)" -> "Add Magnetic Vector Potential" in the boundary condition setting window. The magnetic vector potential boundary condition setting dialog appears (Fig. 2.6-9).

BoundaryCondi	tion	×
Magn	eticVectorPotential	
	Value	
Normal	0.0	
	OK cance	el

Fig. 2.6-9 Boundary Condition Setting Dialog

Please check "Normal" and click "OK".

In this model, since the normal direction component of the magnetic vector potential is set to 0 on the entire surface, do the same setting for all the surfaces.

After adding the boundary conditions, you can check them by selecting "View" -> "Boundary Condition" - > "Cnd Format" in the boundary condition setting window (Fig. 2.6-10).

ConfirmBC	2	×
	sravity 0.0 0.0 0.0 boundary 5 mvpOnFaceGroup 1 0.0 mvpOnFaceGroup 2 1 0.0 mvpOnFaceGroup 2 1 0.0 mvpOnFaceGroup 3 0.0	< >
	] <	>
	ОК	

Fig. 2.6-10 Boundary Condition Confirmation Dialog

After confirming that it is set as shown in Fig. 2.6-10, select "File" -> "Quit" in the boundary condition setting window. When the end confirmation dialog (Fig. 2.6-11) is shown, pressing "OK" automatically saves the set boundary conditions and the boundary condition setting window disappears.

Really Quit?		×
<b>?</b> Really Quit?		
ОК	Cancel	

Fig. 2.6-11 Quit Confirmation Dialog

# 2.6.3 Solver input file creation

Next, select "Analysis" -> "Convert to Input File" and click "OK" in the input file creation dialog (Fig. 2.6-12) to create a solver input file with mesh, boundary conditions, material properties.

🗯 Create Input File	×
Make an input file for solver.	
	OK Cancel

Fig. 2.6-12 Input File Creation Dialog

## 2.7 Analysis Execution

#### 2.7.1 Domain Decomposition

ADVENTURE\_Magnetic reads domain decomposition data based on HDDM as an input. If you select "Analysis" -> "Domain Decomposition", the domain decomposition dialog (Fig. 2.7-1) is shown.

🐲 Domain Decomposition	$\times$
Run domain decomposition.	
You can input domain decompose option. Select what parameter you want to and input value.	set
Input # of subdomains/part	
Input # of elements/subdomain	
(A) # of elements (B) # of parent machines (C) # of subdomains in a parent machine (D) # of elements in a subdomain 13306 (A)	
1 (B) x 66 (C)	(D)
Start Canc	el

Fig. 2.7-1 Domain Decomposition Dialog

Here, based on the number of elements of the mesh, you can specify the size of one subdomain (the smallest unit of the domain handled by one CPU) by one of the following two methods.

- · Number of elements contained in one subdomain (Click "Input # of elements/subdomain")
- Number of subdomains in one part (Click "Input # of subdomains/part")

Since numbers have already been entered so that it moves without problems in the initial settings, please click "Start" as it is this time.

# 2.7.2

# 2.7.3 Solver Execution

Calculation by solver is started. When you select "Analysis" -> "Run Solver", the solver execution dialog (Fig. 2.7-2) is shown. In the solver execution dialog, you can change various analysis condition options provided in the solver. There are four options: "I/O", "Non-linear Magnetostatic Analysis", "Subdomain Solver", and "HDDM Solver".

Solver Execution			×
Execute solver. Please change the option	ns as necessarv		
Subdomain Solver	Time-harmoni	c Eddy Current Analysis	1
1/O HDDM So	lver N	on-linear Magnetostatic i	Analysis
🔲 Maximum Memor	y Usage (MB)		
	1000.0		
		]	
		Start	Cancel

Fig. 2.7-2 Solver Execution Dialog (I/O Option)

# I/O Option (Fig. 2.7-2)

Maximum Memory Usage (MB)

The upper limit of the memory used by each process is n [MByte]. If it exceeds this limit, execution stops at that point.

# HDDM Solver Options (Fig. 2.7-3)

Solver Execution			×
Execute solver. Please change the option	is as necessary.		
Subdomain Solver	Time-harmonic	Eddy Current Anal	ysis
I/O HDDM Sol	ver No	n-linear Magnetos	tatic Analysis
Preconditioner		Tolerance for	Convergence
O (No Preconditioner)	)		
Diagonal Scaling			
			1.0E-5
Tolerance for Dive	ergence	📃 Maximum Nu	Imber
_	1.0510		1000.0
	1.0E10		4000.0

Fig. 2.7-3 Solver Execution Dialog (HDDM Solver)

# Preconditioner

Specify preconditioner to be used for HDDM

- (No preconditioner)
- Diagonal Scaling

## Tolerance for Convergence

Specify the tolerance for convergence of HDDM. This is the relative error of the norm to the residual vector of the residual vector in the HDDM iteration, and it is judged that the HDDM iteration has converged when the relative error becomes smaller than this value.

### Tolerance for Divergence

Specify the tolerance for divergence of HDDM. When the relative error becomes larger than this value, it is judged that the HDDM iteration has diverged and the program is terminated.

# Maximum Number

Specify the upper limit of the number of HDDM iterations. If this value is exceeded, the program will terminate even before convergence is reached.

Subdo	main Solver Time-	harmonio	: Eddy Current An	alvsis	
1/0	HDDM Solver	No	on-linear Magneto	static Analysis	
Form	ulation Method	-	🔲 Non-linear Ite	rative Method	
🖲 A Me	thod		(Not Conside	r Non-linearity)	
🖯 A Me	thod with		Newton Meth	od	
the L	agrange Multiplier		O Picard Metho	bd	
		30.0		0.0	0
🗌 Tol	erance for Divergenc	30.0 e	Characteristi	0.0	0
🗌 Tol	erance for Divergenc	30.0 e 1.0E10	Characteristi	0.01 c Curve	0
Inter Char	erance for Divergenc polation Method for acteristic Curve	30.0 e 1.0E10	Characteristi  v-B Curve v-B <sup>2</sup> Curve Weight for	0.01 c Curve Picard Iteration µ	0
Inter Char	erance for Divergenc polation Method for acteristic Curve First-order Interpolati	30.0 e 1.0E10 on	Characteristi v-B Curve v-B <sup>2</sup> Curve Weight for	0.0 c Curve Picard Iteration µ	1

#### Non-linear Magnetostatic Field Analysis Options (Fig. 2.7-4)

Fig. 2.7-4 Solver Execution Dialog (Non-linear Magnetostatic Analysis)

# Formulation Method

Specify the type of formulation to be used.

- A Method
- A Method with the Lagrange multiplier (subdomain solver is forcibly changed to LU decomposition with pivoting)

## Non-linear Iterative Method

Specifies the type of solution to use as a non-linear iterative method for the magnetic resistivity.

- (Not Consider Non-linearity)
- Newton Method
- · Picard Method

#### Maximum Number of Iterations

Specify the upper limit of the number of iterations of non-linear iteration. If this value is exceeded, the program will terminate even before convergence is reached.

#### **Tolerance for Convergence**

Specify the tolerance for convergence of non-linear iterations. When the error becomes smaller than this value, it is judged that the non-linear iteration has converged.

#### Tolerance for Divergence

Specify the tolerance for divergence of non-linear iterations. When the error becomes larger than this value, it is judged that the non-linear iteration has diverged and the program is terminated.

## Characteristic Curve

Select the characteristic curve to be used with the Newton method. Picard's successive approximation method is used for convergence judgment.

- v-B Curve
- $v-B^2$  Curve

# Interpolation Method for Characteristic Curve

Select interpolation method for characteristic curve with Newton method. Picard's successive approximation method is used for convergence judgment.

- The First-order Interpolation
- The Third-order Interpolation

# Weight for Picard Iterations µ

Specify the weight of  $\mu$  of Picard's successive approximation method.

## Subdomain Solver Options (Fig. 2.7-5)

				×
xecute solver. Please change the optior	ns as nec	essary.		
Subdomain Solver	Time-ha	armonic	Eddy Current Analysis	
I/O HDDM Sol	lver	No	on-linear Magnetostatic A	nalysis
Preconditioner			Accelerative Param	neter for IC
(No Preconditioner	)			
Shifted Incomplete Factorization (IC)	Cholesk	v		1.2
				1.2
Tolerance for Con	wergenc	e	Tolerance for Diver	gence
Tolerance for Con	wergenc	e	Tolerance for Diver	gence
Tolerance for Cor	wergenc	: <b>e</b> 1.0E-9	Tolerance for Diver	gence 1.0E10
Tolerance for Con	wergenc	:e 1.0E-9	Tolerance for Diver	gence 1.0E10
Tolerance for Cor	wergenc	: <b>e</b> 1.0E-9	Tolerance for Diver	gence 1.0E10
Tolerance for Cor	wergenc	: <b>e</b> 1.0E-9	Tolerance for Diver	gence 1.0E10
Tolerance for Cor	wergenc	9 1.0E-9	Tolerance for Diver	gence 1.0E10
Tolerance for Cor	wergenc	1.0E-9	Tolerance for Diver	gence 1.0E10
Tolerance for Cor	wergenc	1.0E-9	Tolerance for Diver	gence

Fig. 2.7-5 Solver Execution Dialog (Subdomain Solver)

#### Preconditioner

Specify preconditioner to be used for linear solver. The following character strings can be specified as *s*.

- (No preconditioner)
- Shifted Incomplete Cholesky Factorization (IC)

# • Shifted Inc Accelerative Parameter for IC

Specify the accelerative parameter for shifted incomplete Cholesky factorization.

#### Tolerance for Convergence

Specify the tolerance for convergence of linear solver. When the error becomes smaller than this value, it is judged that the linear solver has converged.

#### Tolerance for Divergence

Specify the tolerance for divergence of linear solver. When the error becomes larger than this value, it is judged that the linear solver has diverged and the program is terminated.

## Execution of the solver

There is no problem with the default settings, so click on "Start" without changing anything in the solver execution dialog. The solver calculation starts.

The log of solver execution is output to <Documents>\advMagOnWin\ExecSolverForWin.log. Because the line feed code is in UNIX format (LF), please use a software that supports UNIX format line breaks such as WordPad, not Notepad.

#### How to set the number of threads used by the solver

Please refer to Section 1.4.

## 2.8 Result Visualizing

To show the result, use external visualization software such as AVS or ParaView. Therefore, we will export the analysis results to a file for visualization. This time, we will explain the steps to visualize in ParaView.

## 2.8.1 Start Visualizing

If you select "<u>A</u>nalysis" -> "<u>E</u>xport Analysis Result", the export dialog of analysis result (Fig. 2.8-1) is shown. Select "Magnetic Flux Density" and "Electromagnetic Force" as the material quantities to output, "VTK format (such as ParaView)" as the output format, and specify the output folder (Fig. 2.8-1).

🗯 Export Analysis Result		Х		
Select physical quantities and output format destination folder. Multiple physical quantitie	s for exporting, and specify the output is and output formats can be selected.			
Output Physical Quantity	Output Format			
Magnetic Flux Density	UCD Format (AVS)			
Electromagnetic Force	UCD Format (MicroAVS)			
	VTK Format (such as ParaView)			
Output Folder				
	OK Cance	el		

Fig. 2.8-1 Analysis Result Export Dialog

When you click "OK", two files "res.vtu" (magnetic flux density) and "res\_NF.vtu" (electromagnetic force) are output to the selected folder.

If the format is UCD (AVS), avs\_B.inp and avs\_NF.inp will be output, and if UCD (MicroAVS), mavs\_B.inp and mavs\_NF.inp will be output.

If these files exist in the selected folder, the dialog for overwrite confirmation (Fig. 2.8-2) will be shown. Click "Cancel" to return to the original dialog, and specify the output folder again.

Confirm	Overwrite X
	Following file(s) already exist in selected folder. - res.vtu - res_NF.vtu Overwrite?
	OK Cancel

Fig. 2.8-2 Confirm Overwrite Dialog

# 2.8.2 Terminating AdvMagOnWin

The operation of AdvMagOnWin is over at this point.

By saving the analysis case in "<u>File</u>" -> "<u>Save Analysis Case</u>", you can re-analyze later. Exit AdvMagOnWin with "<u>File</u>" -> "E<u>xit</u>".

From this point on, it will be the explanation of ParaView operation.

# 2.8.3 Launching ParaView and Reading Files

After starting ParaView, open the file selection dialog (Fig. 2.8-3) from "File" -> "Open". First, to visualize the magnetic flux density, select "res.vtu" in the output folder and click "OK".

Look in:	C		/		• 🗘	O	0
Examples My Docume Desktop Favorites C:¥ D:¥ Window M/ Documents AdvMagOn	Win-	ename bin dist jie lib sample_data res.vtu res_NF.vtu	Type Folder Folder Folder Folder Vtu File vtu File	Size 814.6 KB 824.9 KB	Date 1 2017/0 2017/0 2017/0 2017/0 2017/0 2017/0	Modif 01/26 01/26 01/26 01/26 01/26 01/27 01/27	ied 17:36: 17:36: 17:36: 17:36: 17:36: 10:48: 10:48:
	File	name: res.vtu					OK

Fig. 2.8-3 Selecting "res.vtu"

You can see that "res.vtu" is selected on the left side of the ParaView main window (Fig. 2.8-4).

III ParaView 5.2.0 64-bit	-		×
<u>File Edit View Sources Filters Tools Catalyst Macros H</u> elp			
😰 🖄 🐯 🕼 🙉 👔 🛃 🛃 🛤 📣 🕨 🕨 🕪 🖾 🎜 Time:💽 💿 🗊			
	<b>9</b> +90 -90	<b>1</b> 2	»
Pipeline Browser 5 × □Layout #1 × +			
■ builtin: #影 30 時 回 图 图 回 图 ● ● 彩 彩 云 云 云 音 Rend	erView1		8 ×
🐵 🤹 res.vtu			
Properties Information			
Properties & X			
C Apply Beset Delete ?			
Search (use Esc to clear text)			
= Properties (re			
Cell/Point Array Status			
- Display			
Axes Grid Edit			
Contar Avas Visibility			

Fig. 2.8-4 Main Window after Selecting "res.vtu"

In this state, the file has not been read yet. Click "Apply" in the "Properties" column on the left side to

show the shape of the analysis model (Fig. 2.8-5).



Fig. 2.8-5 Model Shape Display

## 2.8.4 ParaView 3D screen operation method

ParaView operates the 3D screen as follows.

• Rotation:

Hold down the left mouse button and move the mouse to rotate the model.

Translation

movement::

Hold the mouse wheel button (middle button) and move the mouse to move the model in parallel.

· Zoom:

Turn the mouse wheel upward to zoom in, and turn down to zoom out.

Alternatively, move the mouse upward while holding down the right button of the mouse to zoom out, move to the bottom to zoom in.

#### 2.8.5 Mesh Display

If you change "Representation" in the "Properties" box or "Surface" in the toolbar at the top of the screen to "Surface with Edges", the mesh will be shown (Fig. 2.8-6).

III ParaView 5.2.0 64-bit	_	
<u>File Edit View Sources Filters Tools Catalyst Macros H</u> elp		
😰 🥙 🐯 🙀 \land 🔍 ? 💕 🚉 🕵 🛯 🖉 🕨 🔊 🗈 🕅		
📱 🎴 🛱 🛱 🗟 🕒 Solid Color 🔹 🗾 Surface With Edges 🔹 🔣 📓 🏭 🕌 🕌 🕌	<b>≇1 1</b> ≓ 9	0 » 🥦 »
🖩 👒 🕸 🕸 😒 📥 😒 🚱 🕼 🖳		
Pipeline Browser B × □ Layout #1 × +		
■ builtin:	RenderView1	
Properties Information	À	
Properties 8 ×	ý .	
Coloring		
26		

Fig. 2.8-6 Mesh Display

# 2.8.6 Display of Magnetic Flux Density

If you change "Coloring" in the "Properties" box or "Solid Color" in the toolbar to "MagneticFluxDensity", the magnetic flux density is shown (Fig. 2.8-7).



Fig. 2.8-7 Display of Magnetic Flux Density
To see a specific part, cross section view is effective. To show cross section, select "Slice" (red frame part in Fig. 2.8-7) to enter section setting mode (Fig. 2.8-8).

M ParaView 5.2.0 64-bit	-	
<u>File Edit View Sources Filters Tools Catal</u>	alyst <u>M</u> acros <u>H</u> elp	
🖻 😤 🕼 🗖 🖉	🚉 🐔 📔 🕨 🕪 🛸 🎜 🎜 Time:0	
	🔹 💉 Representation 🔹 🐹 🔣 😳 🗳 🗱 🛱 🗱 🖓 🧬	
	1 🥯 🍘 🖳 📽 💓 🚟 🛞	
Pipeline Browser & ×	□Layout #1 × +	
builtin:	# %   30 ◎   因 因 因 因 跟 强 砷 啦 彩 ◎ 云 云 云 ? 會 <u>RenderView1</u>	
👁 💼 res.vtu		
Ilice1		
Properties Information		
Properties 8 ×		
Search (use Esc to clear text)		tv M
= Properties (SI 🖸 🗈 🚭 🔒 ^		2.60
Slice Type Plane 💌 🔳		
Plane Parameters		.95
Show Plane		201
Origin 0.054999999 0.009550649 0.059999999		.30
Note: Use 'P' to pick 'Origin' on mesh	Y	).65 <sup>°</sup>
or 'Ctrl+P' to snap to the closest mesh point		
X Normal Camera Normal		.350
¥		
	38	

Fig. 2.8-8 After Selecting "Slice"

In this example, we will extract the section near the bottom surface in the Y direction. Set the extraction direction selection button on the lower left to "Y Normal" and move the section (red frame) to the vicinity of the lowest surface in the Y direction (Fig. 2.8-9).

III ParaView 5.2.0 64-bit —	×
Eile Edit View Sources Eilters Tools Catalyst Macros Help	
😥 🤌 🐯 🛱 🍠 ঝ 🤶 💕 🗮 ᇌ 🕅 📣 🕨 🕪 🖬 🛱 Time O	
📲 🚇 🛱 🛱 🛱 🔚 👘 🔺 🗶 🖉 🖉 🛤 🔛 👘 🖉 🖉	»
🖩 📎 🟟 🏟 🗣 😂 😖 🎯 🕼 🗖 📽 🐲 🚟 🕏	
Pipeline Browser & Layout #1 × +	
I builtin:         #影响的意义及图像的影响从人生的         Render/View1         回目	] & ×
👁 💼 res.vtu	
Slice1	
Properties Information	
tr'Apply Beset * Lelete ?	
Search (use Esc to clear text)	tinle
Properties (SI 🕥 🗈 😳 🔔 🛆	0
Slice Type Plane	
Plane Parameters	
Show Plane	
Origin 0.054999995 0.0001 73075 0.059999996	
Note: Use r to plot Origin of mesh	
point El 30er	

Fig. 2.8-9 Setting Cross Section

III ParaView 5.2.0 64-bit	-	$\Box$ $\times$
Eile Edit View Sources Eilters Tools Catalyst Macros Help		
😰 🖄 🐯 🦉 \land 🖓 🛊 🍂 🚺 🛛 🖉 🕨 🖉 Time:OO 🗢		
📘 🎴 🛱 🛱 🗟 • MagneticFluxDensity 🔹 Magnitud 👻 Surface 🔹 🔣 🔀 😳 🗱 🕌 🗱 🗱 🗸	1 📑 🕵	» 🥦 »
🗐 🕲 🕸 🕸 😫 📥 😂 🙆 🚱 🕞 🖳 💥 🕷		
Pipeline Browser 5 × □ Layout #1 × +		
📓 builtin: 🦊 🦛 🛛 🚳 🛛 🕲 🖉 🖉 🕲 🕲 🖉 🖉 🕸 秘 聪 题 🔍 🗛 🗛 🤤 👚 👥 👷	nderView1	
👁 💼 res.vtu		
a 🔨 Slice1		
Glyph		
Properties Information		
Properties 5 ×		
C <sup>R</sup> Apply @ Reset X Delete ?		
Search (use Esc to clear text)		
	neticFluxDensi	ty Magnitude
Properties (Slice D D D C A		.000e+00
Slice Type Plane		05.44
Plane Parameters		.9344
Show Plane	-	30.3
Orgin 0.064999999 0.0001730/96.0699999		
Normal 0 1 0 0 Normal 0 Norma	_	1.65148
X Normal Camera Normal Z Z		
× · · · · · · · · · · · · · · · · · · ·		.316e-05
R.		

When "Apply" is selected, the magnetic flux density of the extracted cross section is shown (Fig. 2.8-10).

Fig. 2.8-10 After Cross Section Extraction

Next, vector the magnetic flux on the section. Clicking on "Glyph" (red frame part in Fig. 2.8-10) results in the view shown in Fig. 2.8-11. If you click "Apply" here, a vector of magnetic flux will be shown with arrows (Fig. 2.8-12).



Fig. 2.8-11 After Selecting "Glyph"



Fig. 2.8-12 Display of Magnetic Flux Vectors

## 2.8.7 Display of Electromagnetic Force

I will also visualize electromagnetic force.

Open "res\_NF.vtu" to show the electromagnetic force vector in the similar process as the magnetic flux density (but change it from "Solid Color" -> "Nodalforce"), as shown in Fig. 2.8-13.



Fig. 2.8-13 Display of Vectors of Electromagnetic Force

#### 3 Time-harmonic Eddy Current Analysis

This chapter explains the procedure of time-harmonic eddy current analysis using the sample data supplied with AdvMagOnWin.

#### 3.1 About the model used

The unit system used in this chapter is SI. It is not necessary to declare a unit system when using AdvMagOnWin. The model used in this chapter is a cake model, which is an accuracy verification model for eddy current analysis using an infinite-length solenoid coil. Fig. 3.1-1 shows this model. The analysis area is the area cut out 20° around the z-axis from Fig. 3.1-1. The radius of the conductor part is 0.1 [m].

The magnetic resistivity v is  $1 / (4\pi \times 10^{-7}) \text{ [m / H]}$  in the whole analysis area, the conductivity  $\sigma$  of the conductor is  $7.7 \times 10^6 \text{ [S / m]}$ , and the angular frequency  $\omega$  is  $2\pi \times 60 \text{ [rad / s]} = 376.9911$ . The magnitudes of the real and imaginary parts of the forced current density J flowing in the coil are 50 and 0 [A / m<sup>2</sup>], respectively. A list of material property values is shown in Table 3.1-1.

Considering the symmetry of the problem, a fan-shaped area with a central angle of 20° and a height of 0.1 m is taken as the model to be analyzed (see Fig. 3.1-2). As boundary conditions,  $A \times n = 0$  and an electric scalar potential  $\varphi = 0$  [V] are imposed on the faces of  $\theta = 0^{\circ}$  (green) and  $\theta = 20^{\circ}$  (red) as shown in Fig. 3.1-3. A is a magnetic vector potential [Wb / m], *n* is a unit normal vector of the boundary surface.



<b>Fig. 3.1-</b>	l Infinite	length	solenoid	coil

Property	Part	Magnetic	Conductivity (S/m)	Volume applied
ID		resistivity		
		(m/H)		
0	Conductor	795774.7	7.7e6	0 (conductor.igs)
1	Inside air	795774.7	Not applicable	1 (air01.igs)
2	Coil	795774.7	Not applicable	2 (coil.igs)
3	Outer air	795774.7	Not applicable	3(air02.igs)

Table 3.1-1 Material property list



Fig. 3.1-2 Cake Model



Fig. 3.1-3 Boundary Conditions

		<b>A</b>
Material	Magnetic Reluctivity	<b>B-H</b> Characteristic
Coil	795774.7	-
Magnetic Material	757.1	-
Air	795774.7	Apply Fig. 2.1-2's characteristic curve

# 3.2 Starting the Software

Please refer to section 2.3.

# 3.3 Creating Analysis Case

This is very similar to the case in Section 2.4, but in "Analysis Type (Subtype)-Electromagnetic Field Analysis", select "Time-harmonic Eddy Current" and click "Next" (Refer to Fig. 3.3-1).

📾 Analysis Type(Subtype) - Electrom	agnetic Field Analysis		_		×
S Start About Analysis Type About Analysis Subtype About Element Type E End	Select Analysis Subtype for: Electromagnetic Field Analysis O Non-linear Magnetostatic Time-harmonic Eddy Current				
		Cancel	< Back	1	vext >

Fig. 3.3-1 Analysis Type (Subtype)

Next, the window in Fig. 3.3-2 appears. Select "IGES" as the geometry model. You can only select "Linear Tetrahedron (Edge Element)" in the analysis model, so just click "Next".

🗯 Element Type		- 🗆 X	
S Start	Select Model Type.		
About Analysis Subtype	Geometry model IGE S	Analysis model Linear Tetrahedron (Edge Element)	
About Element Type	(Mesh)	O Read mesh file	
E End	<ul> <li>Patch</li> <li>ADV_Cad</li> </ul>	○ Tetrahedron	
		Cancel < Back Next >	*

Fig. 3.3-2 Element Type

These operations create an analysis case and display a list of required operations in the procedure guide window. Follow the procedure shown in the procedure guide window and the message window for the

subsequent operations.

Next, we will generate a mesh.

## 3.4 [Preprocess 1] Mesh Generation

### 3.4.1 Selection of CAD Models

The method of specifying a CAD model file as the analysis shape is as described in section 2.5.1. The files to be selected specifically are in the <AdvMagOnWin installation folder>\sample\_data\cake\igs folder.

As shown in Fig. 3.4-1, select conductor.igs, air01.igs, coil.igs, and air02.igs in this order, and click OK to import the IGES files into the analysis case. Table 3.4-1 shows the contents of each IGES file and the area it occupies.



## Fig. 3.4-1 IGES List Window

Volume	IGES file name	Area name	Range of area
ID/Property ID			_
0	conductor.igs	Conductor area	$0 \le r \le 0.1$
1	air01.igs	Inside air area	$0.1 \le r \le 0.15$
2	coil.igs	Coil area	$0.15 \le r \le 0.17$
3	air02.igs	Outer air area	0.17 < r < 0.25

# Table 3.4-1 Information on each IGES files

### 3.4.2 Settings for Node Density

Next, specify the node density. Select "Mesh" -> "Set Node Density" (Fig. 3.4-2).

💐 Settings fo	r Node Density	_		) ×	<
Set node density.					
Base distance					
					1.0
Туре	Range	Intensity of Densi	ty	read file	•
				on Point	t
				on Line	
				on Cylind	er
				remove	
			ок	Cance	el

Fig. 3.4-2 Node Density Setting Dialog

In this analysis example, the node density setting file prepared as example data is read. Click on "read file". When the file open dialog (Fig. 3.4-3) is displayed, select "cake.ptn" in the "<AdvMagOnWin installation folder>\ sample\_data\cake\igs" folder and click "Open".

.ook in:	gs	🖬 🖻 🗖 🔡
🗋 cake.ptn		
ile <u>N</u> ame:	cake.ptn	
ile <u>N</u> ame: iles of <u>Ty</u> pe:	cake.ptn ptn File(*.ptn)	

Fig. 3.4-3 Node Density Setting File Selection

As a result of reading the contents of the file, the basic node spacing is changed to 0.02 m, and two local node density settings of Cylinder type are added (Fig. 3.4-4). Click OK to complete the configuration.

🐲 Setting	s for Node Density	_		×
Set node dens	ity.			
Base distance				
				0.02
Туре	Range	Intensity of Density		read file
Cylinder	0.13,0.1,0.07,0.0,	3.0		
Cylinder	0.19,0.17,0.15,0	2.0	on Point	
				on Line
			01	n Cylinder
				remove
		O	ĸ	Cancel

Fig. 3.4-4 Node Density Setting Dialog after File Selection

## 3.4.3 Create Surface Patch

Next, create surface patches from CAD shapes. If you select "Mesh" -> "Make Patch", the make patch dialog (Fig. 3.4-5) will be displayed. Click "OK" to create a patch.

📾 Make patch	×
Start make surface patches and merge.	
OK Canc	el

Fig. 3.4-5 Make Patch Dialog

## 3.4.4 Mesh Generation

Next, we will generate a mesh. If you select "Mesh" -> "Make Mesh", the mesh generation dialog (Fig. 3.4-6) appears. If you leave the "correct surface patch" checkbox in the dialog as it is, the surface patch is automatically corrected before mesh creation. Click "OK" to begin meshing.

Mesh Generation ×				
Start mesh generation.				
✓ correct surface patch				
OK Can	cel			

Fig. 3.4-6 Mesh Generation Dialog

When mesh generation is complete, the total number of elements and the total number of nodes will be displayed (Fig. 3.4-7). After confirming the number of elements and that of nodes, please click "OK".

Mesh In	nformation	×
i	Total # of elements is 13047 and total # of nodes is 2013	37.
	ОК	

Fig. 3.4-7 Total Number of Elements and Nodes in Mesh File

Then, set the material property values and the boundary conditions.

# 3.5 [Preprocess 2] Setting Analysis Conditions

## 3.5.1 Setting Material Property Values

Material property values are set from "<u>A</u>nalysis" -> "Set <u>Material Property</u>" -> "Electro<u>magnetic Field</u> Analysis" (Fig. 3.5-1).

🦃 Prope	rty Data Settings			×
Set material p volume.	roperty data. When clicking a cell, a di	ialog is	displayed to set the physical p	roperty value of the corresponding
Property ID	Magnetic Reluctivity		Material Property Type	Applied Volume
0	795774.7	Others	0	-3
	·			
			Add Material Properties	Remove Material Properties
Angular Fra				
Angular Fre				270.0011
Angular freq	uency [rad/s] or afternating current i	n coll		376.9911
				OK Cancel

Fig. 3.5-1 Material Property List (Initial)

In the initial screen, one material property with property ID = 0, magnetic resistivity = 795774.7, material property type = other, volumes applied = 0-3 is automatically set. Modify the existing physical property ID and add a new material property ID so that the settings shown in Table 3.1-1 are made.

First, change the material property value of Property ID = 0 to that of the conductor. When the Property ID 0 is clicked, the property change dialog (Fig. 3.5-2) is displayed.

🐲 Material Proper	ty Settings			×
Material Property Ty	Material Property Value Sett Magnetic Reluctivity	lings		795774.7
Magnetic Material	Coll			
O Permanent Magnet	Definition Type		DE	-
Conductor	Deminuon Type		N <sup>2</sup>	
Others	Definition File			
	Magnetic Material			
	Definition File			
	Dormanont Magnot			
	Definition Tuno	DE		_
	Definition Type	rv <sup>-</sup>		· · · · · · · · · · · · · · · · · · ·
	Definition File 1			
	Definition File 2			
Volume to be Applied 0-3	Conductor			
Configurable Volume:0-3	1			
* Applicable volumes can 2-4)	be specified in both range spe	cification and enumer	ation. Combined us	e is also possible (eg: 1,
				OK Cancel

Fig. 3.5-2 Material Property Settings Dialog (Before Editing)

Make the change in the following procedure.

- "Material Property Type"
   Change "Material Property Type" on the left side from "Others" to "Conductor". The "Conductor" setting field is enabled.
- "Conductivity" value Change the "Conductivity" value from "0.0" to "7.6e6".
- 3) "Volume to be Applied"Change "Volume to be Applied" at the bottom from to "0-3" to "0" (Fig. 3.5-3).

After setting change is complete, click on "OK". The material property list dialog is shown again (Fig. 3.5-4).

🗱 Material Proper	ty Settings	×
Material Property Ty	Material Property Value Setting	S
Coil	Magnetic Reluctivity	795774.7
<ul> <li>Magnetic Material</li> </ul>	Coil	
O Permanent Magnet	Definition Type	RF
Conductor	boundari typo	
<ul> <li>Others</li> </ul>	Definition File	
	Magnetic Material	
	Definition File	
	Permanent Magnet	
	Definition Type	RF
	Definition File 1	
	Definition File 2	
	Definition File 2	
	Conductor	
	Conductivity	7.6e6
Volume to be Applied 0		
Configurable Volume:0.3		
* Applicable volumes can	be specified in both range specific	ation and enumeration. Combined use is also possible (ea: 1.
2-4)		
		OK Cancel

Fig. 3.5-3 Property Settings Dialog (After Completion of Conductor Area Editing)

🕷 Property Data Settings 🛛 🗙						
Set material property data. When clicking a cell, a dialog is displayed to set the physical property value of the corresponding volume.						
Property ID	Magnetic Reluctivity	Material Property Type	Applied Volume			
0	795774.7	Conductor	)			
		Add Material Droportion	Domous Material Droportion			
		Add Material Properties	Nemove material Properties			
			376 9911			
			510.3511			
			OK Cancel			
			on ouroor			

Fig. 3.5-4 Material Property Settings Dialog (After Conductor Setting)

Next, set the material property value of the inside air area.

Click "Add Material Properties" to display the "Addition of Material properties" dialog.

For "Material Property Type", leave "Others" specified by default as it is. Enter "795774.7" for "Magnetic Reductivity" and enter "1" for "Volume to be Applied".

Fig. 3.5-5 is in the state where editing is finished. Click "OK" to return to the property data setting dialog.

🐖 Addition of Mat	terial properties					×
Material Property Ty	Material Property Value Settings Magnetic Reluctivity				795774.7	
<ul> <li>Permanent Magnet</li> <li>Conductor</li> <li>Others</li> </ul>	<b>Coil</b> Definition Type Definition File		RF			
	Magnetic Material Definition File					
	Permanent Magnet Definition Type Definition File 1	ē	Æ			•
	Definition File 2					
Volume to be Applied 1	Conductor					
* Applicable volumes can 2-4)	be specified in both rang	e specification and enu	meration. C	ombined use	is also poss	ible (eg: 1, Cancel

Fig. 3.5-5 Material Property Setting Dialog (When Air Area Editing is Complete)

Next, set the coil area. Click "Add Material Properties" and edit the newly displayed dialog as follows.

- "Material Property Type"
   When changing from "Others" to "Coil", the "Coil" setting field becomes enabled.
- 2) "Magnetic Reductivity" Change "0.0" to "795774.7".
- "Coil" -> "Definition type" Change from "RF" to "MD".
- "Coil" -> "Definition File"
   Select "coil.dat" in the "<AdvMagOnWin installation folder>\ sample\_data\cake\done" folder as the "Definition File".
- 5) "Volume to be Applied"Set the "Volume to be Applied" to "2" at the bottom of the dialog.

When all settings are complete, it becomes as shown in Fig. 3.5-6. Click "OK" to return to the Property Data Settings Dialog (Fig. 3.5-7).

💐 Addition of Mat	erial properties	s X
Material Property Ty © Coll O Magnetic Material O Permanent Magnet O Conductor O Others	Material Proper Magnetic Reluct Coil Definition Type Definition File	hty Value Settings tivity 795774.7 MD    J.3b_20190523a\AdvMagOn\Win-0.3b\sample_data\cake\done\coil.dat   monort the file after completing material property settings
	- Magnetic Mater Definition File - Permanent Mag Definition Type Definition File 1	gnet
	Definition File 2	
Volume to be Applied 2	Conductivity	0.0
Configurable Volume:3 * Applicable volumes can 2-4)	be specified in bo	oth range specification and enumeration. Combined use is also possible (eg: 1,

Fig. 3.5-6 Material Property Settings Dialog (After Coil Area Editing is Complete)

olume.	ropeny data. When clicking a cell	, a uraiog is displayed to set the physic	cal property value of the corresponding
Property ID	Magnetic Reluctivity	Material Property Type	Applied Volume
)	795774.7	Conductor	0
	795774.7	Others	1
2	795774.7	Coil	2
		Add Material Properties	Remove Material Properties

Fig. 3.5-7 Property Data Setting Dialog (After Coil Area Setting is Complete)

Next is the setting of the outer air area. I	t is almost identic	cal to the inner	air area setting.	The only
difference is that "3" is specified as "Volum	ne to be Applied"	(Fig. 3.5-8).		

💏 Addition of Ma	terial properties			×
Material Property Ty	Material Property Value Setting	s		
Coil	Magnetic Reluctivity			795774.7
Magnetic Material	03			
O Permanent Magnet	Coll	P		
Conductor	Dennidon Type	rv.		¥
Others	Definition File			
	Magnetic Material			
	Definition File			
	0.01111011110			
	Permanent Magnet			
	Definition Type	RF		-
	Definition File 1			
	Definition File 2			
	Conductor			
	Conductivity			
Volume to be Applied 3				
Configurable Volume:3				
* Applicable volumes can 2-4)	be specified in both range specific	ation and enumeratio	n. Combined use is	s also possible (eg: 1,
				OK Cancel

Fig. 3.5-8 Material Property Settings Dialog (when Editing of the Outer Air Area is Complete)

Finally, set the angular frequency in Fig. 3.5-9. Use the default value of 376.9911 according to the problem settings. Then click "OK".

orunno.			
Property ID	Magnetic Reluctivity	Material Property Type	Applied Volume
	795774.7	Conductor	0
	795774.7	Others	1
	795774.7	Coil	2
	795774.7	Others	3
		Add Material Properties	Remove Material Properties
Angular Fre	luency	Add Material Properties	Remove Material Properties

Fig. 3.5-9 Property Data Setting Dialog (After Settings of All Property IDs are complete)

# 3.5.2 Setting Boundary Condition

Next, set the boundary condition. First, select "<u>A</u>nalysis" -> "Set <u>B</u>oundary Condition", and the dialog shown in Fig. 3.5-10 will be displayed.

	瓣 Set boundary condition	×
	You set boundary conditions. At this time, faces should be identified by classifying surface triangles into groups. If this is your first time or if you failed in extracting a face properly, you select "Extract faces and set boun conditions." and set a face extraction resolution by an angle. If you will not modify previous face extraction, you select "Modify boundary conditions without changing faces."	ndary
	<ul> <li>Extract faces and set boundary conditions. Enter resolution for face extraction as a # of divisions in 180 degrees.</li> <li>180 2.0 = 90.0</li> <li>Modify boundary conditions without changing faces.</li> </ul>	
(a)		el

Fig. 3.5-10 Boundary Condition Setting Dialog

Here, surface triangles (patches) are grouped by their normal directions to extract faces (patch groups) in order to reconstruct CAD topology information lost due to mesh generation. Boundary conditions are added on a patch group basis.

It is necessary to specify a two-sided included angle that corresponds to the resolution of the grouping. The smaller the angle (the larger the value in (a) in Fig. 3.5-10), the finer the surface can be classified. For the current model, there is no problem with the default, so just click "OK". Group division is performed automatically, and the window for setting boundary conditions (ADVENTUR\_BcGUI 2.0) is launched (Fig.



Fig. 3.5-11 Boundary Condition Setting Window (Immediately After Startup)

At this time, the display of the procedure guide window automatically changes from the whole analysis procedure to the boundary condition setting procedure (Fig. 3.5-11 red arrow). Suggestions corresponding to the procedure will also be displayed in the message window, so please proceed according to the suggestions. Refer to section 2.6.2 for how to operate the 3D model on the boundary condition setting screen.

There are two faces to set the boundary condition as shown in Fig. 3.1-3. For example, after selecting the face with  $\theta = 0^{\circ}$ , select "<u>B</u>C" -> "BC (Magnetic)" -> "Add Magnetic Vector Potential" in the menu of the boundary condition setting window (Fig. 3.5-12). Notice also that the selected Surface Group ID is displayed as 1 in the lower status bar in Fig. 3.5-12. The Magnetic vector potential boundary condition setting dialog is displayed (Fig. 3.5-13).



Fig. 3.5-12 Selection of the Menu Item for Boundary Condition Setting After Selecting the Surface with  $\theta = 0^{\circ}$ 

BoundaryCo	ndition	X
Magn	eticVectorPoter	ntial
	Value	
ormal	0.0	
	ОК	cancel

Fig. 3.5-13 Boundary Condition (Magnetic Vector Potential) Setting Dialog

Check "Normal" as indicated by a red circle in the figure, keep the value zero and click "OK". Similarly, set the same conditions for the surface of  $\theta = 20^{\circ}$ . As electric scalar potential  $\varphi = 0$  is a natural boundary condition, setting is not necessary.

After setting the boundary conditions, select "<u>V</u>iew" -> "<u>B</u>oundary Condition" -> "<u>C</u>nd format" in the boundary condition setting window to check the set boundary conditions (Fig. 3.5-14).

Confirm	nBC	×
i	gravity 0.0 0.0 0.0 boundary 2 mvpOnFaceGroup 0 1 0.0 mvpOnFaceGroup 1 1 0.0	^
	<	~
	ОК	

Fig. 3.5-14 Boundary Condition Confirmation Dialog

After confirming that the settings are as shown in Fig. 3.5-14, click the OK button and select the "File"-> "Quit" menu item. When the quit confirmation dialog (Fig. 3.5-15) is displayed, pressing "OK" will automatically save the set boundary conditions and the boundary condition setting window will disappear.

Really Q	uit?	$\times$
?	Really Quit?	
	OK Cancel	

Fig. 3.5-15 Quit Confirmation Dialog

### 3.5.3 Solver input file creation

Then select "<u>A</u>nalysis"-> "<u>C</u>onvert to input file" and click "OK" in the input file creation dialog (**Fig. 3.5-16**). A solver input file is created that merges meshes, boundary conditions, and material property values<sup>1</sup>.

<sup>&</sup>lt;sup>1</sup> To be precise, only the file that describes the relationship between volume ID and material property ID is included in the solver input file. There is no need to worry about that as long as you are using the Windows version.

🖮 Create Input File	×
Make an input file for solver.	
	OK Cancel

Fig. 3.5-16 Input File Creating Dialog

- 3.6 Analysis Execution
- 3.6.1 Domain Decomposition

ADVENTURE\_Magnetic reads domain decomposed data based on the HDDM as input. Select "<u>A</u>nalysis"-> "<u>D</u>omain Decomposition", to display the domain decomposition dialog (Fig. 3.6-1).

🐲 Domain Decomposition	×
Run domain decomposition.	
You can input domain decompose option. Select what and input value.	parameter you want to set
Input # of subdomains/part	
Input # of elements/subdomain	
(A) # of elements (B) # of parent machines (C) # of subdomains in a parent machine (D) # of elements in a subdomain 13047 (A)	
$1(B) \times 65$ (C)	(D)
	Start Cancel

Fig. 3.6-1 Domain Decomposition Dialog

Here, based on the number of elements of the mesh, the size of a subdomain (the smallest unit of the domain handled by one CPU) can be specified by one of the following two methods.

- · Number of elements contained in a subdomain (Click "Input # of elements/subdomain")
- Number of subdomains in a part (Click "Input # of subdomains/part")

The recommended values have already been entered automatically so that domain decomposition can be performed without problems, so just click on "Start".

#### 3.6.2 Solver Execution

Next is the start of the calculation by the solver. Select "<u>A</u>nalysis"-> "<u>R</u>un Solver" and the solver run dialog (Fig. 3.6-2) will be displayed.

🗯 Solver Execution		×
Execute solver. Please change the options as necessary.		
Subdomain Solver Time-harmonic Eddy Current #	\nalysis	
1/0 HDDM Solver Non-linear Magne	tostatic Ana	lysis
Maximum Memory Usage (MB)		
1000.0		
	Start	Cancel

Fig. 3.6-2 Solver Execution Dialog (I/O option)

In the solver execution dialog, you can change various analysis condition options provided by the solver. The options are divided into four parts: "I/O", "HDDM Solver", "Non-linear Magnetostatic Analysis", "Subdomain Solver", and "Time-harmonic Eddy Current Analysis".

**I/O Option (Fig. 3.6-2)** Please refer to Section 2.7.2.

## HDDM Solver Options (Fig. 2.7-3)

Please refer to Section 2.7.2.

# **Time-harmonic Eddy Current Analysis Option (Fig. 3.6 3)** Refer to Fig. 3.6-3.

濊 Solver Execution				~
Solver Execution				
(ecute solver. Ilease change the option	ns as neress:	anz		
Subdomain Solvor	Timo harmo	nie Eddy Curront A	nakeje	
1/0 HDDM So	ver	Non-linear Magnet	tostatic An	alvsis
Eormulation Metho	d			
	4			
• A-•				
O A Method				

Fig. 3.6-3 Solver Execution Dialog (Time-harmonic Eddy Current Analysis Option)

## Formulation Method

Specify the type of formulation to be used.

- A Method
- · A- $\phi$ Method

# Subdomain Solver Options (Fig. 2.7-5)

Please refer to Section 2.7.2.

# **Execution of Solver**

There is no problem with the default settings, so click on "Start" without changing anything in the solver execution dialog. The solver calculation starts.

The log of solver execution is output to <Documents>\advMagOnWin\ExecSolverForWin.log. Because the line feed code is in UNIX format (LF), please use a software that supports UNIX format line breaks such as WordPad, not Notepad<sup>2</sup>.

<sup>&</sup>lt;sup>2</sup> Notepad on Windows 10 October 2018 Update (1809) now supports LF.

# 3.7 Result Visualization

The way to display the results is the same as described in section 2.8. Here we will explain only the differences from Section 2.8. Also, the description of the visualization method by MicroAVS and AVS/Express is omitted.

# 3.7.1 Start of Visualization

If you select "<u>A</u>nalysis" -> "<u>E</u>xport Analysis Result", the Export Analysis Result Dialog (Fig. 3.7-1) will be displayed. Of the physical quantities to be output, "Electromagnetic Force" cannot be selected because it is not used in this analysis type. Select "VTK Format (such as ParaView)" as the output format, and specify the output folder (Fig. 3.7-1).

🗯 Export Analysis Result	×
Select physical quantities and output format destination folder. Multiple physical quantitie	s for exporting, and specify the output is and output formats can be selected.
Output Physical Quantity	Output Format
🗹 Magn. Flux/Eddy Curr. Density	UCD Format (AVS)
Electromagnetic Force	UCD Format (MicroAVS)
	✓ VTK Format (such as ParaView)
Output Folder 0190523a\AdvMagOnWin-0.3	3b\sample_data\cake\cakeFull_vis
	OK Cancel

Fig. 3.7-1 Export Analysis Result Dialog

When "OK" is clicked, nine files, i.e., "res.vtu<sup>3</sup>" (magnetic flux density and eddy current density, VTK), "avs\_Bi.inp" (magnetic flux density (imaginary part), AVS/Express), "avs\_Br.inp" (magnetic flux density (real part), AVS/Express), "mavs\_Bi.inp" (magnetic flux density (imaginary part), MicroAVS), "mavs\_Br.inp" (magnetic flux density (real part), MicroAVS), "avs\_Jei.inp" (eddy current density (imaginary part), AVS/Express), "avs\_Jer.inp" (eddy current density (real part), AVS/Express), "mavs\_Jei.inp" (eddy current density (imaginary part), MicroAVS), and "mavs\_Jer.inp" (eddy current density (real part), MicroAVS) will be output.

# 3.7.2 Terminating AdvMagOnWin

Please refer to section 2.8.2.

## 3.7.3 Launching ParaView and Reading Files

In this section, we used version 5.6.0 of ParaView for Windows (64 bit). After launching ParaView, open the file selection dialog (Fig. 3.7-2) from "File"-> "Open". To visualize the magnetic flux density, select "res.vtu" in the output folder specified in section 3.7.1 and click "OK".

<sup>&</sup>lt;sup>3</sup> The vtu extension corresponds to unstructured grid data in VTK format.

Examples	Filename	/ Tune		_
My Documents	aux Rijon	ion File		
Desktop	avs_binp	inp File		
Favorites	avs_biinp	inp File		
C:¥	avs_tering	inp File		
D:¥	mays Bijon	inp File		
Q;¥	mays_bitinp	inp File		
R:¥	mays_billion	inp File		
Windows Network	mays_jet.inp	inp File		
	inavs_yetanp	utu File		

Fig. 3.7-2 Selecting "res.vtu"

You can see that "res.vtu" is selected in the left part of ParaView's main window (Fig. 3.7-3).

III ParaView 5.6.0 64-bit
<u>File Edit View Sources Filters Tools Catalyst</u>
6 🖗 🕅 🗱 🖉 🕐 🛛
Pipeline Browser 🗗 🗙
Properties   Information
Properties 🗗 🛪
Apply     Reset     Delete     ?       Search (use Esc to clear text)     Image: Search (use Esc to clear text)     Image: Search (use Esc to clear text)       Properties (r     Image: Search (use Esc to clear text)     Image: Search (use Esc to clear text)
Cell/Point Array Status     Set EddyCurrentDensity-Imag     Set EddyCurrentDensity-Real     Set MagneticFluxDensity-Imag     MagneticFluxDensity-Real

Fig. 3.7-3 Main Window after Selecting "res.vtu"

The file has not been read yet in this state. Click "Apply" in the "Properties" tab on the left to display the shape of the analysis model (Fig. 3.7-4).



Fig. 3.7-4 Showing the Model Shape

# 3.7.4 Method of 3D Screen Operation of ParaView

See Section 2.8.4.

# 3.7.5 Displaying Mesh

If you change "Representation" in the "Properties" tab or "Surface" (Fig. 3.7-6) in the toolbar at the top of the screen to "Surface with Edges", the mesh will be displayed (Fig. 3.7-6).

III ParaView 5.6.0 64-bit <u>File Edit View Sources Filters Tools Catalyst Macros Help</u> 🔎 🔍 🗗 🚉 況 🛛 🔍 見 Time: 0 🛱 🛱 🙀 👬 🕒 Solid Color • Surface 6 r. 🕀 📚 🥒 ᅇ M Pipeline Browser ₽× □ Layout #1 × | + | 🤗 % | 3D 🔞 | 🔣 🔣 🔣 | 🖳 🖳 🕸 🕸 🕸 🗟 🛝 🗛 🗛 🦓 🖉 💼 📋 builtin: res.vtu

Fig. 3.7-5 Switching to Mesh Display



Fig. 3.7-6 Mesh Displayed

# 3.7.6 Displaying Magnetic Flux Density

If you change "Coloring" in the "Properties" tab or "Solid Color" in the toolbar to "MagneticFlux Density-Imag" (Fig. 3.7-7), the imaginary part of the magnetic flux density is displayed (Fig. 3.7-8).

III ParaView 5.6.0 64-bit							
<u>File Edit View Sources Filters Tools Catalyst Macros H</u> elp							
] 🖻 🤔 🐯 🐯 🙆 🖉 🦉							
	Solid Color						
🛛 🗐 🜍 🧊 🦃 🚱	EddyCurrentDensity-Imag     EddyCurrentDensity-Real						
Pipeline Browser	MagneticFluxDensity-Imag						
builtin:	<ul> <li>MagneticFluxDensity-Real</li> <li>I I III III IIII IIII IIIIIIIIIIIIIII</li></ul>						
⊛ 💼 res.vtu							





Fig. 3.7-8 Display of Magnetic Flux Density Imaginary Part

The color map is different from the commonly used one, so let's change it. With the data whose color you want to adjust is selected in the tree, click "Edit Color Map" (red circle in Fig. 3.7-10).



Fig. 3.7-9 Location of "Edit Color Map"

To make detailed settings for "Edit Color Map", click the gear icon 🕮 (red circle in Fig. 3.7-10).



Fig. 3.7-10 Gear Icon

Click the icon P to change the color map (Fig. 3.7-11).



Fig. 3.7-11 The Icon for Changing Color map

Selecting "jet" (red ellipse 1) in the "Choose Preset" dialog (Fig. 3.7-12) will display the most commonly used color map. After clicking on "Apply" (red ellipse 2) in the lower right group of buttons, close the dialog with "Close" (red ellipse 3).



Fig. 3.7-12 Preset Color Map Selection Dialog

Clicking on the button to the right of "Color Map Editor" closes the "Color Map Editor" and the main screen looks like Fig. 3.7-13.

III ParaView 5.6.0 64-bit			- 0 ×
File Edit View Sources Filters Tools Catalyst	Macros Help		11 m m
📲 🏊 🚜 🛱 🛱 🛣 💿 Magnet	ticFluxDensity-Imag 💌 Magnitude 💌 Surface With Edges	I X X Q X X X X X Z Z Z Z Q 2 I Z C	» makeAnimeRM20_100 *
I 🖩 🕥 🗭 🛱 🏟 🎯 🏈 😂 🏄	③ ℃ ] □		
Pipeline Browser & X	□ Layout #1 ×   +		
builtin:	<i>勇</i>	P Res	nderView1 □□□♂×
Properties Information Properties Status Properties Status Search - (use Ecs to clear text) Properties (res.vtu) Properties (res.vtu) Cell/Point Array Status Cell/Point Array Status Cell/Point Array Status Search - (use EddyCurrentDensity-Imag Search - (use EddyCurrentDensity-Imag			●prtudbom
			2

Fig. 3.7-13 Imaginary Part of Magnetic Flux Density after Color Map Change

Next, draw a vector diagram, but before that, change the whole model to wireframe display. Select "Wireframe" from the pull-down in Fig. 3.7-14.



Fig. 3.7-14 Selecting Wireframe

When the wire frame display is displayed, click "Glyph" indicated by a red circle in Fig. 3.7-15.



Fig. 3.7-15 Selecting Glyph while Displaying in Wire Frame

Select "MagneticFluxDensity-Imag" again in "Orientation Array" or "Scale Array". For this result, for example, "Scale Factor" was set to 30,000. Press "Apply" and the vector will be displayed (Fig. 3.7-16).



Fig. 3.7-16 Vector Diagram (Imaginary Part of Magnetic Flux Density)

Similarly, the real part of the magnetic flux density (Fig. 3.7-17), the imaginary part of the eddy current density (Fig. 3.7-18), and the real part of the eddy current density (Fig. 3.7-19) are shown. For these, "Blue to Red Rainbow" was adopted as a color map.

/// ParaView 5.6.0 64-bit		– 0 ×
<u>Eile Edit View Sources Filters Tools Catalyst M</u> acros <u>H</u> elp		10 m - m
⊘ 🔗 🐯 😳 🔊 🕫 🐨 🚱 🛛 🗠 🕨 🕨	🛚 🕵 🛛 Time: 🛛 👘 🖉 🖓	
📕 🎴 🍰 🛱 🛱 👬 🎲 🔿 MagneticFluxDensity-Real 💌 Magnitude	Surface 💽 🔣 💥 🗔 💥 🕯	× 1+Y -Y1 +Z1 1-Z C 90 90 12 00 >> makeAnimeRM20_100 >>
🔲 🕥 🛱 🛱 🖗 🗣 🧟 📥 😒 🐻 🔲 🌌 🐲 🎬	8	
Pipeline Browser		
🖥 builtin: 🥖 👯 🕺 🐯	■ 国 ■ 単 略 略 哀 ▲ ヘ ? 2 言	Render View1 🔲 🖻 🗗 🗗 🖈
🔹 💼 res.vtu		
Givph1		
Properties Information   Properties		90 - 1.3e-06 gg
Orientation Array O MagneticFluxDensity-Real		
Scale		e e
Scale Array		
Vector Scale Mode Scale by Magnitude		56-7
Scale Factor		
		Ê.
Glyth Made		
Maximum Number		- 1.10-12 0
Of Sample Points 5000		100
Seed Innoon		2
	14	

Fig. 3.7-17 Vector Diagram (Real Part of Magnetic Flux Density)

Protections Mode Scale Detertion - Image - Ima	III ParaView 5.6.0 64-bit File Edit View Sources Filters Tools Catalyst Macros H		- 0 ×
Protecties Information Properties Information	🖻 👌 🕅 🕅 🖒 🍠 🖎 🗗 🚉 💱		
Properties Information Properties Information Proper	EddyCurrentDensity	mag V Magnitude V Surface V X 1 1 X 1 1 X 1 + Y - Y 1 + Z 1 1 - Z C 2 1 1 - Z	» makeAnimeRM20 100 »
Properties Information Properties Information			-
Properties Information   Properties Informa			
Properties Information Properties Information Properties Properties State Properties State Properties State Resert. (use East to clear text) Crientation Array © EddyCurrentDensity-Imag ♥ Scale Factor • 0 80013 ♥ Masking dipth Mode Information Name of Samele Properties 1000 Seed The State Distribution ♥ Masking State Distribution ♥ Seed Factor • 1 80013 ♥ Masking State Distribution ♥ Seed Factor • 1 80013 ♥ Seed	builtin:	□ Layout #1×  +     # %   30 個   型 図 型   図 図 陣 敏 秘 総 図 ▲ △ ? 2 音 <u>Rer</u>	nderView1 🛛 🖂 🗗 🗸
Properties Information   Properties Information   Properties Information Array © EddyCurrentDensity-Imag Scale Array © EddyCurrentDensity-Imag Scale Array © EddyCurrentDensity-Imag Scale Factor : 100 1001 Maxking Maxking Maxking Scale Distribution Maxing Maxking Scale Distribution Scale Factor : 100 1001 Maxing Bioph Mode Maxing Scale Distribution Scale Factor : 100 1001 Maxing Scale Distribution Scale Table : 100 1001 Scale Distribution Scale : 100 1001 Scale : 1001 1001 1001 Scale : 1001 1001 1001 Scale	<ul> <li>m res.vtu</li> </ul>		
Properties Information   Properties	Glyph1		
Properties Information   Properties Properties Formation   Properties			
Properties Information Properties To formation Properties To class the second s			
Properties Information   Properties			
Properties Information   Properties			
Properties Information Properties For the set to clear text The s			l,
Properties Information Properties Caloba Constraints of the second secon			
Image: Construction of the sector of the	Properties Information   Properties EX		
Search - (use Ect to clare text)         Orientation Array         Scale         Scale Array         Octor Scale Mode         Scale Factor         D0013         Array         Scale Factor         Scale Factor         Octor Scale Mode         Scale Factor         Octor Scale Mode         Scale Factor         Octor Scale Mode         Scale Totor         Octor Scale Mode         Scale Factor         Octor Scale Mode         Scale Totor         Scale Totor         Scale Totor         Scale Totor         Scale Totor         Scale Totor			
Seede       Scale       Maximu <ul> <li>42e+01</li> <li>9</li> <li>42e+01</li> <li>9</li> <li>9</li></ul>	Preset Delete		eg d
Orientation	Search (use Esc to clear text)		ji contra
Circle Marking       Scale fractor       - 30         Glyph Mode       Uniform Spatial Distribution       - 30         Masking       - 10         Glyph Mode       Uniform Spatial Distribution         Vector Scale Points       - 0.0e+00         Scale       - 0.0e+00         Scale       - 0.0e+00         Scale       - 0.0e+00         Glyph Mode       - 0.0e+00         118339       - 0.0e+00	Orientation		- 4.2e+01 p
Scale	EddyCurrentDensity-Imag		20
Vector Scale by Magnitude Scale Factor - 00013 - 00013 - 00013 - 00013 - 00010 - 000+000 - 000+00-00-00-00-00-00-00-00-00-00-00-00-	Scale Array		- <sup>30</sup> E
Scale Factor (Scale by Magnitude Masking Glyph Mode Masing Number Seed 1039	Vector Scale Mode Costs by Magnitude		20 → Ž
Masking:     Image: Constraint of the second s	Scale Factor		eus
Preskute     Uniform Spetial Distribution     Image: Constraint of the second s		ť.	_ 10 윧
Maximum Number Of Sample Points         5000	Glyph Mode Uniform Spatial Distribution	l≟_⊀	
Ut Sample Points 1 <sup>1111</sup> Seed 118399 I	Maximum Number 5000		ر 10+00.0 – ح
	Seed 10339		d
			ш

Fig. 3.7-18 Vector Diagram (Imaginary Part of Eddy Current Density)
M ParaView 5.6.0 64-bit				– ø ×
File Edit View Sources Filters Tools Catalyst Macros Hi				
6 🖗 🗱 🛱 🖑 🔊 🔍 💣 🗮 🔗	I I I I I I I I I I I I I I I I I I I	0 🛨 🛛 🚳 🖕		
📕 🎴 🍰 🚞 🛣 🔁 👬 🐻 🔹 EddyCurrentDensity	-Real 💌 Magnitude 💌 🗍 Surface	- X X Q + 1	+Y -Y1 +Z1 1-Z ♀ ♀ ↓ ↓ ⓒ »	makeAnimeRM20_100 *
6` 62 🖕 🧟 🖗 🦉 🕼 🧳 🧑	] 📭 🤮 💓 🎬 🛞 🥒			
Pipeline Browser & ×	□ Layout #1 ×   +			
builtin:	🤗 %   30 🚳   🕅 📉 🗮   📉 🔣 🕸 🕸	🕅 🐴 🐴 🥝 🖉 🏢	Render	rView1 🔲 🖯 🗗 🗡
Tes.vtu				
Glyph1				
😄 🗕 💼 Glyph2			1	
Glyph3				
			AT A A AKIN	
			A A A A A A A A A A A A A A A A A A A	
		A AN		
			TXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX	
			THAT ALK W	
Properties Information	5.24	- TERRETORIA - VIIII		
Properties & X	- The			
Panok Recet Pelete 2	A			
			MARKER AND	<u>a</u>
Search (use Esc to clear text)		Contraction of the	A A A A A A A A A A A A A A A A A A A	ğ
Orientation 🔺				3.8e+01 🛱
Orientation Array				Ja
Scale				- 30 등
Scale Array				<u>Š</u>
Vector Scale Mode October March 1				20
Vector Scale Mode Scale by Magnitude				insi-
Scale Factor				_ 10 🖉
Masking				e
Glyph Mode Uniform Spatial Distribution				ظ مميون م
Maximum Number 5000				
Seed 10839				qq
	)			

Fig. 3.7-19 Vector Diagram (Real Part of Eddy Current Density)

# 4 Useful Functions

# 4.1 Mesh Extraction of Selected Volume

From the mesh consisting of multiple volumes, create a partial mesh consisting of nodes and elements contained in some selected volumes.

After generating the mesh, select "Mesh" -> "Extract Selected Volume", the mesh extraction dialog (Fig. 4.1-1) of the selected volume will be shown.

Mesh Extraction of Selected Volume		
Create a partial n	nesh that extracts only elements belonging to the selected volume.	
Select volumes.	Output File   **** About output file name ***   The following 4 files are output by an extraction processing.   1) mesh file (*.msh)   2) node number conversion table file (*.src_node)   3) element number conversion table file (*.src_elem)   4) volume number conversion table file (*.src_vol)   For each file name, the selected file is followed by an extension.	
		OK Cancel

Fig. 4.1-1 Volume Selection Window

Select the volume you want to extract and click the button to the right of the input field of "Output File", the save dialog (Fig. 4.1-2) will be shown. Please specify the save destination directory, enter the prefix of output files and click "Save". However, saving of the file is not completed yet.

🗰 Save			×
Save In:	AdvMagOn	Win-0.2b	• a d d 885
📑 bin		🗋 advmagonwin_en.bat	
📑 dist		iAgent.conf	
🗂 doc		log.properties	
📑 jre			
📑 lib			
📑 sample_da	ata		
advmagon	win.bat		
File <u>N</u> ame:			
Files of <u>Type</u> :	All Files		<b>•</b>
			Save Cancel

Fig. 4.1-2 File Saving Destination Selection Dialog

Since you will return to the volume selection screen again, please click "OK" when you confirm that the file name is entered in the "Output File" column. This completes saving, and the four files "<file name>.msh", "<file name>.src\_node", "<file name>.src\_elem", and "<file name>.src\_vol" are output. When these files exist in the selected folder, the dialog for overwrite confirmation (Fig. 4.1-3) will be shown. If you want to overwrite, click "OK". Otherwise, click "Cancel" and return to export dialog.

Confirm	Overwrite	×
	Output files exist. Over	write to it?
	OK Cancel	

Fig. 4.1-3 Confirm Overwrite Dialog

As an extraction example, Fig. 4.1-4 shows the visualization result of the mesh extracted only volume 0 from the mesh created in Chapter 2.



Fig. 4.1-4 Result of Visualization of Extracted Mesh

#### 4.2 Creation of B-H curve definition file

You can compose the relationship (B-H curve) between the magnitude of magnetic field strength H [A / m] and the magnitude of magnetic flux density B [T], which is the material property of the magnetic material, while viewing the plot result and save it as a file. The B-H curve is defined by specifying two or more sizes of B (|B|) and H (|H|) and defining the segmented curves connecting these points.

### 4.2.1 Startup

When you select "Analysis" -> "Set Material Property" -> "Create B-H Curve Definition File" with AdvMagOnWin activated, the B-H curve definition file creation window (Fig. 4.2-1 is shown).



Fig. 4.2-1 B-H Curve Definition File Creating Window

The plot area of the B-H curve is shown on the left side of the screen. On the right side there are a list of defined values of the B-H curve, four buttons for the definition value manipulation function, and a button for closing the creation window.

#### 4.2.2 Adding definition values

When you click "Add Value", the Add Item dialog (Fig. 4.2-2) is shown, so enter the value of the definition point of the B-H curve (Fig. 4.2-3). Please input the size of H (| H |) and the size of B (| B |) respectively.

ОК	Cancel
IBI	0.0
H	0.0
述 Add Item	×



Fig. 4.2-2 Add Item (Initial)

Fig. 4.2-3 Add Item (After Input)

Click "OK" to return to the original screen, the specified point is shown at the center of the graph on the left

side, and the coordinate value of the point specified on the right side is shown (Fig. 4.2-4). The scale of the graph changes depending on the size of the specified value.



Fig. 4.2-4 After Adding Definition Point

If you add another point, the points added to the graph are also shown, and a line is drawn between the points that originally existed (Fig. 4.2-5). The table on the right side of the screen is always sorted automatically with the value of |H|.



Fig. 4.2-5 Show 2 Added Points

#### 4.2.3 Delete definition value

To delete an unnecessary definition value, select the row of the definition value you want to delete and click "Delete Selected Value" to delete it (Fig. 4.2-6)



Fig. 4.2-6 After Deleting The Definition Point (1.0, 2.0)

## 4.2.4 Reading the definition file

To read the B-H curve definition file of the format used for ADVENTURE\_Magnetic, click "Read File" and select the file in the file selection dialog.

Fig. 4.2-7 shows the state immediately after reading the sample data bh\_curve file.



Fig. 4.2-7 After Reading Sample Data "bh\_curve"

### 4.2.5 Saving the B-H curve definition file

By clicking "Write File", you can save the created B-H curve definition in a file (Fig. 4.2-8)

🖮 Save	×
Save In:	shaft 💌 🖬 🗇 🗗 📴 🗁
test_shaft	12.files 🗋 mtrl.dat
🗋 air.igs	🗋 shaft.ptn
bh_curve	test_shaft12.iag
🗋 coil.dat	
Coil01.igs	
Coil02.igs	
🗋 mag.igs	
File <u>N</u> ame:	
Files of <u>Type</u> :	All Files 🔍
	Save Cancel

Fig. 4.2-8 Save Dialog

When the file with same name exists in the selected folder, the dialog for overwrite confirmation (Fig. 4.2-9) will be shown. If you want to overwrite, click "OK". Otherwise, click "Cancel" and return to original screen.

Confirm	Overwrite	×
$\wedge$	Output file exists. Overwrite to	o it?
	OK Cancel	

## Fig. 4.2-9 Confirm Overwrite Dialog

#### 4.2.6 Creation complete

Click "Close" to finish creating the B-H curve definition. Before closing, please check whether it saved in the file.