# Electromagnetic coupled with Acoustics Tutorials for NX-Magnetics

### Dr. Binde Ingenieure

#### January 2, 2024

2011-2025 Dr. Binde Ingenieure, Design & Engineering GmbH. All Rights Reserved. This software and related documentation are proprietary to Dr. Binde Ingenieure, Design & Engineering GmbH. All other trademarks are the property of their respective owners.

DR. BINDE INGENIEURE, DESIGN & ENGINEERING GMBH MAKES NO WARRANTY WHATSOEVER, EXPRESSED OR IMPLIED THAT THE PROGRAM AND ITS DOCUMEN-TATION ARE FREE FROM ERRORS AND DEFECTS. IN NO EVENT SHALL DR. BINDE INGENIEURE, DESIGN & ENGINEERING GMBH BECOME LIABLE TO THE USER OR ANY PARTY FOR ANY LOSS, INCLUDING BUT NOT LIMITED TO, LOSS OF TIME, MONEY OR GOODWILL, WHICH MAY ARISE FROM THE USE OF THE PROGRAM AND ITS DOCUMENTATION.

THIS SIMULATION SOFTWARE USES FINITE ELEMENT METHODS. USERS SHOULD BE AWARE THAT RESULTS CAN HAVE UNPREDICTABLE ERRORS IF INPUT DATA IS NOT COMPLETELY CORRECT. THEREFORE ANY DESIGN DECISIONS SHOULD NOT BE BASED SOLELY ON THE SIMULATION. USE ADDITIONAL MEASUREMENTS TO ENSURE THE CORRECTNESS.

## Contents

1	Intr	oduction	3
<b>2</b>	Aco	ustics of Transformer with Housing	4
	2.1	Create Model for Magnetic Forces	4
	2.2	Fourier-Transform Magnetic Forces	10
	2.3	Create Model for Acoustics	12
	2.4	Postprocess Acoustic Results	19

### 1 Introduction

This guide shows problems with electromagnetic acoustics coupling. The electromagnetic part is solved using the Magnetics solver. The acoustics part is solved by Simcenter Nastran.

### 2 Acoustics of Transformer with Housing

In this example we want to analyse for the acoustics that result from magnetic forces on a transformer. There are three main steps to do:

First, in a time domain analysis we compute the magnetic forces that act on the transformer and its housing. In the second step, we transfer those forces into frequency domain by using a solution type that provides fourier transformation (FFT). In the last step we perform a Simcenter Nastran, vibro-acoustic simulation with the forces from magnetics.



Estimated time: 1.5 h.

Follow the steps:

### 2.1 Create Model for Magnetic Forces

- download and unzip the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/15.CouplAcoustics/15.1TrafoHousing. zip
- 2. Start Simcenter, click Open 🖻 and navigate to folder 'start'. Select the file 'TrafoHousingAcoustic.prt' and click OK.
- 3. Check the bodies and the air volume around the housing.
- 4. Start application Pre/Post and click 'New Fem and Sim (Non-Manifold)' from toolbar



Magnetics

- 5. Change the displayed part to the Fem part.
- 6. Check the non-manifold faces: In 'Groups', check that there is a group named 'nonmanifold face'. Select the group and verify that the correct faces highlight. Such faces show the interfaces (matings) between two bodies. Here, the mesh must be conformal (identical nodes).



- 7. Blank the AIR\_EMAG and the HOUSING polygon bodies for easier visibility and selection.
- 8. Use the tetrahedral mesher with following settings.
  - Set the 'Type' to 'Tetra10\_compatible'. Hint: For the Magnetics solver, this makes no difference. But this setting allows to use this mesh as a Tetra10 mesh if the Nastran solver runs. Thus, we can use these meshes for both solvers now.
  - with activated 'Automatic Element Size' and
  - 'ElementSizeFactor' = 1 to mesh the bodies in the following order (small parts first). Hint: We use a coarse mesh to speed up the process. Later, to find precise results, the 'ElementSizeFactor' can be set to 0.5 or 0.25.

3D Tetrahedral Mesh	0?X
✓ Objects to Mesh	
✓ Select Bodies (1)	÷
✓ Element Properties	
Туре	Tetra10_compatible
<ul> <li>Mesh Parameters</li> </ul>	
Automatic Element Size	
Element Size Factor	ElementSizeFactor=1
Surface Maximum Growth Rate	1.3 🔹
Surface Meshing Method	Standard 👻
Mech Quality Ontions	

- 9. Mesh the bodies and set the materials as shown in brackets.
  - CORE (Material: 'MU3: Relative permeability 1000')

- COIL1, COIL2 (Material: 'Copper: 5.77e7 Siemens/meter')
- FOOT1, ..., FOOT7 (Material: 'Aluminum: 3.8e7 Siemens/meter')
- HOUSING (Material: Material: 'Aluminum: 3.8e7 Siemens/meter')



- 10. Finally, unblank the AIR\_EMAG polygon body and mesh it in the same way.
  - Put this mesh into the existing mesh collector 'AIR (3d)'.

		Name
		TrafoHousingAcoustic_fem1.fem
		+ 🗇 TrafoHousingAcoustic_fem1_i.prt
Volume iviesn Settings		🔹 🗹 🗁 Polygon Geometry
		Mesh Controls
Model Cleanup Options		+ 📝 💠 2D Collectors
· · · · · · · · · · · · · · · · · · ·		- 🖌 🖉 3D Collectors
- Destination Collector		🖃 🖌 🧱 AIR (3d)
<ul> <li>Destination Collector</li> </ul>		AIR_EMAG
		- VIII CORE
Automatic Creation		CORE
		E COIL1
Mesh Collector	AIR (3d) 👻 🌇	COIL1
		e COIL2
Devenden v Mandan		Solid(1)
Boundary Nodes	P	FOOT6
01/		
ОК	Apply Cancel	- VE HOUSING
		- May Housing

- blank all meshes for easier visibility.
- 11. Define the coils:
  - Edit the Fem file and in 'Geometry Options', activate 'Splines'.

	FEM Name	O Geometry Options 0? X				
	▼ CAD Part					
Name		<ul> <li>CAD Geometry to Include</li> </ul>				
TrafoHousingAcoustic fem1.fem	Idealized Part					
Trafallausing Age durate	▼ Bodies	Points				
	Edit Padias to Usa	Create Mesh Points				
🗄 🗸 🗁 Polygon Geor 🗛 🛛 Freeze Mo		Coordinate Surtains				
Mach Control	Strategy to Use Non Manifo	Coordinate Systems				
V age Mesh Control	<ul> <li>Non-manifold Ontions</li> </ul>	Lines     Arcs/Circles     Solines				
🗄 🗹 🗇 2D Collectors 🔋 Pack						
The AB Collectors	Tolerance 0.01					
The solution of the solution o						
CSYS						
🛛 🖓 🔂 Selection Reci 😽 Display Sin	<ul> <li>Geometry</li> </ul>	Conics				
		Sketch Curves				
⊕ 🔮 Groups	Geometry Options					
+ Fields		Datum Planes				

• Edit the 'Mesh Associated Data' of both coil meshes, set the 'Method' to 'Tangent Curve' and for each coil, select the spline as shown in below picture.

COIL1	♦ Mesh Associated Data
	✓ Select Mesh (1)
COIL2 🗘 Show Tiny Edges	Element Properties
COI 🔂 Lock	Electric Current Direction  Method Tangent Curve
Solid(1)	Primary Direction
FOC FOC	Reset to Defaults
FOC 🥖 Edit Display	✓ Mesh Properties
- 🗹 🕼 FOC 🅼 Edit Mesh Associated Data	Export Mesh to Solver
	Preview Show Result

- Edit Physical property of COIL1 and COIL2.
  - Set the 'Conductor Type' to 'Stranded',
  - the 'Coil Section Area' to  $120^*15~\mathrm{mm}$
  - $-\,$  the 'Number of Turns' to 90
  - the 'Fill Factor' to 0.8
- 12. Set the displayed part to the Sim file.
- 13. create a expression (shortcut Strg+E) for the main frequency that we want to use. Name it 'EM\_Freq', set the 'Dimensionality' to 'Frequency' and use 300 Hz as formula.

ø	Expr	essions					
		† Name	Formula	Value	Units	Dimensionality	1
	1	➤ Default Group					
	2	EM_Freq	300	300	Hz 🔻	Frequency 🔹 🔻	Ν
	3	- ande	🔒 ug_var("angle")	1	•	Angle	Ν

- 14. Create a new Solution for the Magnetics solver. Solution Type is 'Magnetodynamic Transient'
- 15. in register 'Output Requests', 'Plot', activate 'Nodal Force entire (virtual)'. Hint: This is the necessary output for the following fourier-transformation and acoustic simulation. Other results can be activated also if desired.

Solution		<b>ა?</b> ×	<ul> <li>Magnetodynamic Tr</li> </ul>	ransient
<ul> <li>Solution</li> </ul>	[]		···· Output Requests ···· Time Steps	✓ Plot ✓ Magnetic Fluxdensity
Name	Magnetics		Initial Conditions	Magnetic Fieldstrength
Solver	MAGNETICS	•	- Coupled Thermal	Electric Fluxdensity
Analysis Type	3D Electromagnetics	•	- Coupled Elasticity	Electric Fieldstrength
Solution Type	Magnetodynamic Transient	•	Coupled Particle	Current Density
Reference Set	Entire Part	Ŧ		Magnetic Potential (a-Pot)
	Create Solution			Electric Potential (phi-Pot)

16. in register 'Table', activate 'Total Force - entire (virtual)' to allow post processing of these.

Output Requests	Plot
Time Steps	▼ Table
Initial Conditions	
Coupled Thermal	✓ Total Force - entire (virtual)
	Total Moment - entire (virtual)

- 17. in register 'Time Steps',
  - set the 'End Time Option' to 'End Time'
  - and the 'End Time' to '1/EM\_Freq'.
  - set the 'Time Increment' to '1/EM\_Freq/12'. Thus, we will have 12 time steps going over one period depending on the defined frequency. Hint: This is a coarse time step. For higher accuracy, make it smaller.
    - Magnetodynamic Transient

Output Requests	Time Increment	1/EM_Freq/12	s • •
Time Steps	End Time Option	End Time	•
Initial Conditions	End Time	1/EM Erog	
- Coupled Thermal	End fime	1/Elvi_Freq	s • •

- 18. Click Ok to finish the solution window.
- 19. Edit the 'Solver Parameter'. In register 'General', set the 'Result File Type' to 'ascii'. Hint: the result will be written in readable ascii format now. This is necessary for the following fourier-transformation. As a disadvantage, the reading and writing of results will be slower as in the default binary setting.

~	Solver Parameters		ა? X
	Solver     MAGNETICS		
	✓ Parameters		
	General	Solver Version	1169, Build Date 2023/05/
	Cluster	Description	<u>L</u>
Magnetics	Parameter Sweep	Result Tables (txt)	Overwrite 👻
Edit	Parameter Import	Result Graphs (afu)	Create, delete txt Files 🔻
Cor Edit Spiver Parameters	User Defined	Result Plot File Result File Type	Create

20. Create a constraint of type 'Flux Tangent' on all 6 outside faces of the air volume.



- 21. On each coil, create a load of type 'Current'.
  - Use the default type 'On Stranded Coil'
  - Select the Physical: COIL1 and COIL2 for the second.
  - Set the 'Method' to 'Harmonic (cos)',

- 'Electric Current Amplitude' to -15 A
- 'Frequency' to 'EM\_Freq'

Phase Shift' to 90 deg.			
Magnetics			
Result New Load	]		
Current	<b>ა?</b> X		
left Stranded Coil	•		
▼ Name			
Current(1)			
Description			
Destination Folder			
✓ Stranded Coil			
Type SolidPhysical	•		
✓ Select Physical ⊖ COIL1::TlafoHo	- 0		
✓ Magnitude			
Method Harmonic (cos)	-		
Electric Current Amplitude -15	A • •		
Frequency EM_Freq	Hz • •		
Phase Shift 90	° • •		
Card Name CurrentOnStranded3D			

- 22. Save the Sim file
- 23. Solve the Magnetics solution. The solve time will be approximately 5 min.
- 24. Postprocess the results if desired.
- 25. The following picture left shows the total forces in z on the core and on the housing. The right picture shows the Magnetic Fluxdensity in the core (Post View set to Combine at Elements).

TrafoHousingAcoustic\_sim1 : Magnetics Result Load Case 1, Increment 4, 0.0011s Magnetic Flux Density - Element-Nodal, Element-Value, Average, Magnitur Min : 0.000. Max : 1.263. Units = T



### 2.2 Fourier-Transform Magnetic Forces

1. from toolbar 'Acoustics and Vibration' (maybe this toolbar is invisible, thus, make it visible first), Click on 'Model and Load Pre-processing'. A new solution will be created and visible in the Simulation-Navigator.



2. Use the Windows file manager to rename the Magnetics results file-extension from 'bun' to 'unv'.

Hint: Our magnetics result file is now formatted with ascii type (we have requested that above). Normally, such files are then named with 'unv' extension (universal file) but our file has still the extension 'bun' (binary universal file). Thus, we must rename it for further usage. Maybe you must unload it in the postprocessor to allow Windows to rename it. Making a copy of the file also works.



- 3. In the solution 'Model and Load Processing', 'Add a Load' for the 'Input File'.
- 4. in the file browser, select the Magnetics results file (extension 'unv')
- 5. in box 'Unit System', set 'Length' to mm.
- 6. click the 'Refresh' button and verify that the data set 'Force' is found.

7. click OK.

			O Input File	2					ა?
			Operation Name						
			Input File 1						
			🔻 Data So	urce					
			15.0	CouplAcoustics <sup>1</sup>	15.1TrafoHousing	\start\TrafoHousing	Acoustic_sim1-l	Magnetics.unv	f
			👻 Unit Sys	tem					
			The values	in the file are in	terpreted with the	se units			
			Length					mm	-
			Mass					kg	-
			► Identifie	rs					
			Mech Source						
			Iviesii so     Toto	Jurce					
			▼ Data						
			Refresh						C
Magnetics			Data Sets						
Model and Load Pre-processing 1			Selected	Quantity	Name	Location	Sorting	Target Storage Name	
🕺 🗄 Add Load	• 🕑	<sup>7</sup> Input File	~	Force	Force	Nodes	Vectors	FORCE-RESULTS	
🕮 Add Operation	• 🗄	Tumulent						01	
	R	ranourcitu						ОК	Cancel

8. There is another unit setting that must also be changed: Edit the solution 'Model and Load Processing' and set the 'Output File Units' to (mN)(mm)(Kg)

		Solver Parameters	<u> </u>
		Name	
hodel and Load Pre	e-processing 1	Model and Load Pre-processing 1	
🖹 🎢 Input File 1	🔊 Add Operation	✓ Output File Units	
🗏 🐻 Time Signal Pro		Units (Force)(Length)(Mass) (mN)(mm)(kg	g) 🔻
	G Edit	Specify Temperature Unit	

9. In the solution 'Model and Load Processing', click 'Add Operation' and select 'Time Signal Processing'.

	援 Mesh Mapping
Magnetics	Fan Noise Segmentation
Model and Load Pre-processing 1	ղև, Time to Waterfall of Time
Add Operation	Time Signal Processing

10. Set the properties as shown in the below picture. Click OK. ♥ Time Signal Processing ♥ ? ×

Operation Name				
Time Signal Processing 1				
<ul> <li>Input Range Selection</li> </ul>	<ul> <li>Fourier Transform</li> </ul>	n		
Lower Limit Type None	🗹 Enable			
	Window Type	Hanning		-
Upper Limit Type None 👻	Correction Mode	Amplitude		•
	Advanced			
Signal Repetition	<ul> <li>Fourier Transform</li> </ul>	n Output Post	-processi	ng
Number of Repetitions	Lower Limit	30	Hz	•
	Upper Limit	2000	Hz	• •
Time Segmentation		L		
✓ Fourier Transform	<ul> <li>Average Spectra</li> </ul>	after Segment	ation	

11. Solve the solution 'Model and Load Processing'. The solve time is about 3 min.

Magnetics		5 Solution Monitor -
Model and Load Pre-pro	Add Operation	Stop         Graphs         Inspect         Keep Open         Find
		TrafoHousingAcoustic_sim1-Model_and_Load_Pre-processing_1.log
Time Signal Processing	🖉 Edit	איני האהעי אראה אור
	• Rename	
	₩ Clone	xxx x
	🧼 Validate	X X X0000X X0X X000X XX X000X X0000X X0000X X0X X XX X
	Solve	

- 12. after the solve has finished, verify in the Windows file manager that there is a new file generated with extension 'results.sc\_h5'.
  - TrafoHousingAcoustic\_sim1-Model\_and\_Load\_Pre-processing\_1.bat
  - TrafoHousingAcoustic\_sim1-Model\_and\_Load\_Pre-processing\_1.log
  - TrafoHousingAcoustic\_sim1-Model\_and\_Load\_Pre-processing\_1-config.sc\_h5
  - TrafoHousingAcoustic\_sim1-Model\_and\_Load\_Pre-processing\_1-config.xml

TrafoHousingAcoustic\_sim1-Model\_and\_Load\_Pre-processing\_1-results.sc\_h5

#### 2.3 Create Model for Acoustics

- 1. In the Sim file, create a new solution,
  - Name it 'Acoustics'
  - set the solver to Simcenter Nastran
  - set the 'Analysis Type' to 'Vibro-Acoustic'
  - set the 'Solution Type' to 'SOL 108 Direct Frequency Response' Hint: Other solution types are also possible.

Solution	ა?	_×
▼ Solution		
Name	Acoustics	
Solver	Simcenter Nastran	•
Analysis Type	Vibro-Acoustic	•
Solution Type	SOL 108 Direct Frequency Response	•
Reference Set	Entire Part	Ŧ
	Create Solution	

- click 'Create Solution'
- in register 'Bulk Data', in box 'Fluid-Structure and Poro-Elastic Interface', create a Modeling-object for 'Fluid-Structure Interface Modeling Parameters'

General	FEM Adaptive Order - FEMAO	
File Management	Parameters (PARAM)	
Executive Control	- Eluid Structure and Dava Electic Interface	
Case Control	Fluid-Structure and Poro-Elastic Intenace	<u></u>
Bulk Data	Fluid-Structure Interface Modeling Parameters	None 👻 🐴

• set the 'Type of Coupling' to 'Effect of Structure on Fluid Only (WEAK)'

	g raianeters	
Modeling Object		
✓ Properties		
Description		L.
Card Name	ACMODL	
Type of Coupling (CTYPE)	Effect of Structure on Fluid Only (WEAK)	•
Search Units (SRCHUNIT)	Relative	-

 in register 'Case Control', in box 'Additional Options', create a Modeling-object for 'Fluid-Structure Interaction Control Parameters' and accept all the defaults.

						· · · ·
				Modeling Object		
Case Control	✓ Output Requests	Vibro-Acoustic Outpu	- 8	✓ Properties		
Bulk Data	Structural Damping Parameters	None	- 4	Description		[⊉
				Symmetric Request (ACSYM)	YES	•
	Fluid Damping Parameters	None	•	Particle Velocity Output Type (ACOUT)	PEAK	•
	✓ Additional Options		Reference Pressure (PREFDB)	MPa	• •	
	Parameters (PARAM)	None -	<b>@</b>	Coupled Solution (ASCOUP)	YES	•
				Coupling Information (SKINOUT)	FREEFACE	-
	User Defined Text	None 👻	2	Output Coupling Matrix to Punch File (AGGPCH)	NO	•
	Global Glue Parameters	None 👻	2	Import Coupling Matrix From SFE AKUSMOD (SFEF70)	NO	•
	Fluid-Structure Interaction Control Parameters	None 🗸	2		ОК	Cancel

•	Click OK to crea	te the solution. د د د د د د د د د د د د د د د د د د د		
	✓ Solution			
	Name	Subcase - Direct Frequency 1		
	Solver Type	Simcenter Nastran		
	Analysis Type Vibro-Acoustic			
	Solution	SOL 108 Direct Frequency Response		
	Step	Subcase - Direct Frequency		
		Create Step		

- A new window 'Solution Step' appears. Accept the default step 'Direct Frequency'. click 'Create Step' and OK.
- 2. Create a constraint of type 'Fixed' on one of the foot faces.

-		
- 🗸 📥 Temperatures	;	
Simulation O	bjects	
🗸 🚽 Constraints		
- 🚰 Forcing Frequer	X Delete	
🖻 🖶 Subcase - Direc	New Constraint	Muse Differed Constan
🗸 🛃 Loads	1.ª New constraint	Ger Derined Constra
🔤 📅 Forcing Free	New Constraint Set 🕨	Part Enforced Displacement
🗄 🦳 Results	I≣ Page ►	Selocity
📲 Model and Load Pre	processing i	Enforced Acceleration
🖃 🍸 Input File 1		Fixed Constraint
🗄 🐻 Time Signal Proc	cessing 1	

- 3. Create a load of type 'Force from External File'
  - use the file browser and select the newly created file with extension 'results.sc\_h5'
  - click the button 'Infer Data Set'
  - click OK and the load is generated.

		Force from Exter	nal File	0 ? X
		Name		
		Destination Fold	der	
		✓ Model Objects		
		Group Reference		
		Body Focus		
		🗸 Select Object (0)		ф …
🗑 Acoustics		Excluded		
Simulation Objects		▼ Data Source		
Constraints	Acceleration	✓ Select File		
	📥 Force	V sinel Mardal	Land Land Dec analysis 1	and the set of
📲 Forcing Frequencies	🐴 Force from External File		Land_Load_Pre-processing_1-	-results.sc_no
🗄 🖶 Subcase - Direct Frequency 1	는 Moment 년	Degrees of Freedom	Translation	•
	Rearing	Subcase	From Solution Step	•
Forcing Fr 🗡 Delete	v∰r bearing	Forces Data Set	FORCE-RESULTS	
🕂 🗁 Results 🛃 New Load	😁) Torque			Infer Data Set
Model and Load	📥 Pressure			

4. Create Forcing Frequencies

Recoustics	Modeling Object	
Temperatures	▼ Properties	
Simulation Objects	Description	[
Constraints	▼ Frequency List	
Fixed(1)	Frequency List Form Linear Sweep Defined by	Linear Sweep (FREQ1)
E Subcase - Direct Frequency 1	Start Frequency	20 Hz 👻
Forcing Frequencies	End Frequency After	1000 Hz 🔻
Constant Results     Constant Requestion     Results     Constant Requestion     Results     Results     Constant Requestion     Results     Resu	Step Value	10 Hz •

Forcing Frequencies

**ა?** X

- 5. Prepare the Fem part for acoustics
  - Change the displayed part to the Fem file
  - Edit the Fem part and set the solver to Simcenter Nastran and the 'Analysis Type' to 'Vibro-Acoustics'

Edit FEM	ບ ? X
FEM Name	
▼ CAD Part	
Idealized Part	
Bodies	
✓ Geometry	
Geometry	Options
✓ Solver Environment	
Solver	Simcenter Nastran 👻
Analysis Type	Vibro-Acoustic 🔹

• Deactivate the air-mesh that was used for electromagnetics: Edit the 'Mesh Associated Data' and deactivate the option 'Export to Solver'

	"	Mesh Associated Data	<b> </b>
Simulation Navigator		✓ Mesh	
Name Trafo Houring Acoustic for	45 Edit Mesh Associated Data 교회 Rename	✓ Select Mesh (1)	<del>\</del>
TrafoHousingAcoustic	imes Delete	<ul> <li>Element Properties</li> </ul>	
	👌 Check	Electric Current Direction     Method None	<b>•</b>
🗉 🗹 💠 2D Collectors	Solid Properties		
□ ↓ 3D Collectors	(i) Information	Reset to Defaults	
🖃 🖌 📑 AIR (3d)	🗋 Solver Syntax Preview	<ul> <li>Mesh Properties</li> </ul>	
		Export Mesh to Solver	

- blank all meshes, blank also the AIR\_EMAG polygon body. Unblank all other polygon bodies.
- 6. Create the acoustics mesh
  - from toolbar 'Home', group 'Mesh', select 'Acoustic Mesh Automation'

పి 3D Tetrahedral 양 2D Mesh की Mesh Control	▲ 項 /	More	≪ ₩	More	Auton
Mesh		Recent	tly Used	1	
🍬 육 🖵 🔻 👒		Aco	ustic Me	sh Autom	ation

- drag a window to select all polygon bodies (not the AIR\_EMAG)
- set the 'Element Size' to 10 mm at both 'Acoustic Shell Surface Wrap' and 'Convex Mesh'
- set the 'Offset Distance' to 50 mm.

Acoustic Mesh Automation	ა <b>?</b> X		
✓ Selection			
Select Objects (11)	<b>⊕</b> …		
✓ Acoustic Shell Surface Wrap			
Element Size 10	mm 🕶 🏓		
Hole Closing Size 1	mm • •	- Solid From Shall M	lach
Smoothing Level			esn
	0.5000	<ul> <li>Element Properties</li> </ul>	
Snap To Source Boundaries		Туре	CTETRA(4) - Acoustic Fluid
<ul> <li>Destination Collector</li> </ul>		Internal Mesh Gradation	1.05
✓ Convex Mesh		🗌 Target Internal Edge l	Length Limit
Element Size 10	mm 🔻 🔻	▼ Destination Collector	or
Offset Method Distance	Along Normal 🔹		Character 1
Offset Distance 50	mm 🔻 🔻		Show P
Infinite Plane			OK Apply (

• click OK and the mesh is created. Assign material Air to it.

PSOL	ID - Acoustic Fluid	<b>ర?</b> ×
▼ Phys	ical Property Table	
Name	PSOLID - Acoustic Fluid1	
Label	8	
▼ Prop	erties	
Material	Air	•
	ОК	Cancel

• edit the Material 'Air' and disable 'Bulk Modulus' and 'Bulk Modulus Ratio'.

Fluid Material			ა? ×
✓ View			
MAT10			•
MATT10			
✓ Name - Description			
Air			
Label			11
Description			
Pedigree			
Check Properties based on co	urrent Property View		
<ul> <li>Categorization</li> </ul>			
▼ Properties			
Mass Density (RHO)	1.2041		kg/m³ → =
Damping Coefficient (GE)			=
Mechanical	✓ Properties		
	Bulk Modulus (K)		MPa •=
	Bulk Modulus Ratio (GAMMA)		=
	Speed of Sound (C)	343.21	m/s 🔹 💌

• blank the acoustic air mesh and also the 2D meshes that have been created.

- 7. Create an Automatically Matched Layer (AML)
  - Create a Simulation Object of type 'Automatically Matched Layer'.

🗉 📲 Magnetics		Automatically Matched Layer	ა? x
		▶ Name	
Iemperatures      Imperatures      Imperatures		Destination Folder	
+ V Constraints	$\times$ Delete	<ul> <li>Automatically Matched Layer Surf</li> </ul>	faces
Forcing Frequencies	[⊕] Select All	Automatically Matched Layer Surface	None 🔻 🇞 🔻
+ Ł Loads	🔩 Remove All Simulation C	Radiation Surface	Automatically Matched Layer Surface 🔻
🗊 Surface-to-Surface Gluing	🗞 New Simulation Object	Automatically Matched Layer Sett	tings
Edge-to-Surface Gluing	(i) Information	✓ Infinite Planes	
🔏 Edge-to-Edge Gluing	🖓 Filter	Infinite Plane 1	None 🔻 🇞 🔻
Panel	<b>≜</b> ↓ Sort	Infinite Plane 2	None 👻 🔹 🔻
Acoustic Absorber	Eind Object	Infinite Plane 3	None 🔻 🇞 🔻
🍓 Automatically Matched Layer	P This objection	Card Name AMI REG	

• click 'Create Region' and set the selection filter to 'Feature Angle Element Face' and select one of the outside element faces of the acoustic air mesh. All others will be selected. Click OK twice to create the condition.

▼ Name				F	eature Angle Eleme 👻 🗌
Name	AmlRegion1				K-X-X-
Label	1				
✓ Region C	Dbjects	_	PV		
Group Re	ference				$\mathbf{X} \times \mathbf{X}$
Body Fo	cus				A X X
Ӿ Select OI	oject (0)	<del>ф</del> [			Element Face : (262501)
Excluded	i -				
Card Name E	3SURFS			X	
	ОК	Cancel			
					XN

- 8. Create microphone meshes
  - from toolbar 'Nodes and Elements', group 'Elements', create a 'Plane'

Mesh Primitives	
🚸 Plane 📐	Sphere
+++ Point Set	😭 ISO Power (ISO3744)
🗇 Box Surface	纉 Box Solid

• set the parameters as shown in the picture for the first plane and click OK.

	017
<ul> <li>Mesh Name</li> </ul>	
Mesh Name	Plane Mesh Primitive (1)
✓ Element Prope	erties
Туре	CQUAD4 - Microphone 🔹
<ul> <li>Primitive Parar</li> </ul>	meters
Method	At Center 👻
🗸 CSYS	🛃 🛃 - 🖊
Offset X	0 mm • •
Offset Y	0 mm
Offset Z	0 mm
Length X	2000 mm • •
Length Y	2000 mm * *
<ul> <li>Mesh Paramet</li> </ul>	ters
Maximum Element	Size 50 mm 👻 🗲
<ul> <li>Destination Co</li> </ul>	ollector
Automatic Creat	tion

• do the same for the two other orientations.



- 9. change the displayed part to the Sim file.
- 10. There are some errors in the mesh, which would not allow us to solve the solution. Therefor we can either fix those errors by using smaller element sizes or we can disable the mesh check from the solution. We are going to use the second option, since decreasing the element sizes exponentially increases the time it takes to solve.

To disable the error check, edit the solution and change the 'Geometry Check' in 'Executive Control' to 'None'.

Solution		ა? X
<ul> <li>Solution</li> </ul>		
Name	Acoustics	
Solver	Simcenter Nastran	
Analysis Type	Vibro-Acoustic	
Solution Type	SOL 108 Direct Frequency Response	
Reference Set	Entire Part	Ŧ
✓ SOL 108 Dire	ect Frequency Response	

			Preview
General	Geometry Check	NONE	
- File Managemer	Max Job Time		
- Executive Contro			
Case Control	User Defined Text	None	<u> </u>
Bulk Data			

11. solve the acoustic solution. The needed time is about 1.5h.

#### 2.4 Postprocess Acoustic Results

- 1. In the post-processor, open the result from the acoustic simulation
- 2. choose the frequency 300 Hz and show the result 'Acoustic Pressure'
- 3. Edit the Postview and in 'Result', activate 'Apply dB Scaling'
- 4. Measuring the sound pressure spectrum at one point
  - Click on 'Create Graph'
  - set it to 'Across Iterations'
  - select one node at a location where we want to measure the sound pressure. Choose for instance a location with a high pressure.
  - click OK. The graph will be created. We can see that there are peaks at the main frequency 300 Hz and multiplications of it 600, 900 Hz. Values below 0 dB mean the sound is below the reference and not possible to hear.



• the below picture shows the plot of the acoustic pressure.



The tutorial is finished. Save your parts and close them. References:

- Simcenter Nastran Acoustics User's Guide
- Simcenter Nastran Quick Reference Guide