CAD package for electromagnetic and thermal analysis using finite elements

Flux by CEDRAT

User guide Flux® 11.2

New features

CEDRAT
design solutions for electrical engineering

This user guide (new features) was published on 9 December 2013

Réf. : K101-112-EN-12/13

CEDRAT
15 Chemin de Malacher - Inovallée
38246 MEYLAN Cedex
France

Téléphone : +33 (0)4.76.90.50.45
Fax : +33 (0)4.56.38.08.30

Email : cedrat@cedrat.com
Web : http://www.cedrat.com

CONTENTS

1. Introduction of new features of Flux 11.2 version

Introduction

This chapter introduces the **new features** of **Flux 11.2 version**.

Contents

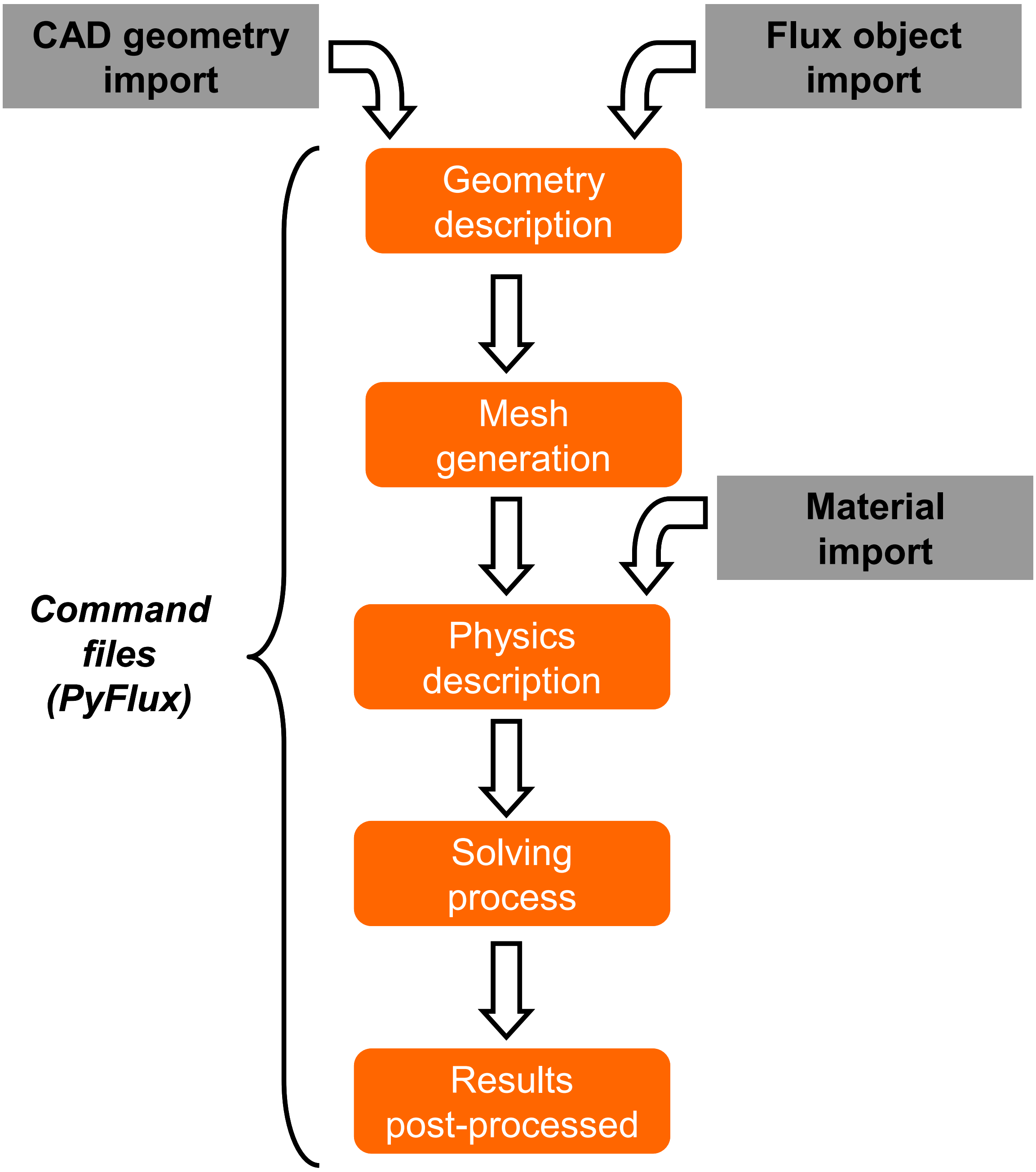
This chapter contains following topics:

1.1. V11.2 new features: list of main new features

Introduction This chapter presents a list of the main new features of **Flux 11.2 version**.

The main news features are listed and the chapter references are given, where the necessary information for a good usage of the new software capabilities is presented in detail.

To localize the impact of the various new features, the flowchart of Flux software principle is presented below.



Continued on next page

Environment / Ergonomics

This item introduces the new features concerning the environment and the ergonomics.

New features	More details in…
The **supervisor ergonomy** has been modified fully. In particular, this new supervisor allows **facilitating**: • the **Flux access** • the **examples access** • the **edition and execution of PyFlux command files**.	§ 2.1
A **python editor** is now available in Flux. The old "**PyFlux command**" window is divided in three parts : • On the extreme right, are **log .py files** to save command performed by the user during a flux session or flux project. • On the middle, an **editor python** zone to open, read and create script. • On the extreme left, a **"Command Prompt"** zone.	§ 2.2
It is possible to modify the look of the Flux window on the screen, i.e.: • modify the background color • display / hide certain zones • resize (reduce / enlarge) zones	§ 2.3
To manage the license, two tools are available for the user : • New tool : The **license viewer** is used to display the authorized applications, authorized versions, the expiration date, the total number of tokens used ... • Improvement of tool **License manager** : is used to configure the license server and connect it Flux to the license system.	§ 2.4

Geometry

This item introduces the new features concerning the geometric preprocessor.

New features	More details in…
The **new import formats** are : • **SOLIDWORKS** (extension :*.SLDASM, *.SLDPRT) • **PARASOLID** (extension :*.PRT) • **NX** (extension : *.X_T, *. XMT_TXT, *.X_B, *.XMT_BIN) These formats are available only for the import, and not for the export. These formats are not available with Linux OS. These formats are available for the import in « advanced mode ».	§ 3.1

Continued on next page

Geometry (continued)

The sketcher is a « drawing » tool that facilitates easier planning and execution of CAD 2D geometries. The integration of a sketcher into Flux software carried out in the version V11.1 brings about an easier and better performing description of the geometry in order to be similar to the current, globally used CAD tools. The version V11.2 of Flux **consolidates the sketcher** environment, bringing about **improvements to existing functions**, as well as **adding new ones**, as example: • Import of image on background • Manage geometric defects • Manage "curve" lines • Graphic improvements…	§ 3.2

Physics

This item introduces the new features concerning the physics description.

New features	More details in…
The anisotropic materials have a behavior law which depends on the field direction. Anisotropies can be of different type : • the **crystalline anisotropy**, connected with the crystalline structure of the material which originates from the manufacturing procedure • the **pseudo-anisotropy** linked to lamination and modeled by the homogenization of a package of stacked sheet metal This section presents new information about Flux software that takes into consideration these two forms of anisotropy : • a **new type of region** (**Laminated magnetic non conducting region)** to model the pseudo-anisotropy linked with lamination • **new material models** to model the crystalline anisotropy in 2D	§ 4.1
User sub routines allow extending the scope of application of Flux. They provide the opportunity to define new physical properties (sources, materials,…) based on settings chosen by user. Thus, user is able to model non-standard physical phenomena. In V11.2, ser sub routines are written in **Groovy** (no need compiler and user version).	§ 4.2
The Bertotti model allows computing the iron losses after the solving, for steady state and transient application. In V11.2, the user has the possibility to **adjust** more **coefficients** of the theorical formula, the **exponents**.	§ 4.3

Continued on next page

Solving process

This item introduces the new features concerning the solving process.

New features	More details in…
The **adaptive solver** is a process that permits modeling a problem by means of an **adaptive mesh**. This mesh presents sizes of elements, which correspond to **the local behavior of the considered physics**. In the zones requiring a finer analysis of the studied phenomenon, this translates by: • a tightening of the meshes • a diminution of the finite elements size	§ 5.1
In V11.2, the transient initialisation has been improved. The transient initialisation allows fixing the initial conditions. • by static computation (already existing in Flux) • by **finite elements solution** (V11.1 new feature + improvement and finalization in V11.2)	§ 5.2
In the framework **parallel calculus** technology deployment **MUMPS** was integrated in Flux with two distinct versions : • **Mumps : direct solver** • **Mumps distributed (beta version)**	§ 5.3
In the continuity of **improvement works about Flux Skew** achieved in V11.1, the version 11.2 of Flux some points about this application: • Infinite box capability has been added • Geometic parameter is now available • Display of volume regions after solving • 3D geometry building speed increased prior to the solving process • «Stop / Resume solving » management	§ 5.4
A substantive work has been performed in order to improve the Flux solver and especially for problems of no convergence. The solving of projects with circuit has been greatly improved.	

Continued on next page

Vibro-acoustic coupling

This item introduces the new features concerning the magneto vibro-acoustic coupling.

New features	More details in…
The **magneto-vibro-acoustic coupling** facilitates the export of magnetic forces or harmonics according to the time variation of these forces, which are result of the transient magnetic Flux application. These results are then used with other software packages carrying out the vibro-acoustic portion of the computations, like **LMS-VL** or **Nastran**. These analyses vary in the study of the function of the device (motor, actuator, ...). Such functionality exists in the 2D, Skewed and 3D models, with or without symmetries and periodicities.	§ 6

Material manager

This item introduces the new features concerning materials.

New features	More details in…
The **new material manager** is an interface facilitating materials management in a database. It comprises two working contexts: • Editing of databases: it permits the user to add, modify, import or delete materials and databases • Consultation of the databases: it allows the user to see and compare the information on materials and to compare the curves.	§ 7
An **Excel sheet** permitting to **determine coefficients of flux models of soft materials (2 coefficients and 3 coefficients)** is available. This sheet allows **fitting an analytic model with experimental data**. This tool has been stored on our space disk, during the installation, at the following path : *C:\Cedrat\FluxDocExamples_11.2\Tools\FitMaterialAnalyticModelsWithExperimentalData*	

Distribution / Optimization

This item introduces the new features concerning the distribution of computation and the optimization of a study

New features	More details in…
Cedrat Distribution Engine is a software component enabling distributed computing in Cedrat applications. **Cedrat Distribution Engine** enables the distribution of **Flux** computations run from the optimiser **GOT-It**.	§ 8

1.2. V11.2 new features: macros

Macros

This item presents the V11.2 new features concerning the new macros since the 11.1.0 version.

Context	New macros
2D Physics	**CreateCoilWithSolidConductor.PFM** Create Coil with each turn defined as a solid conductor.
2D Postprocessing	**CreatePathFromLineRegion.PFM** Create one or two compound path(s) from a line region.
	RunFrozenPermeability.PFM Solve again an already solved scenario, using relative permeability for each step of solving scenario.
	CreateSensorFor2DSlotForce.PFM Create sensors to compute force on slots for motors.
	BertottiIronLossesVsSlipAcIm.PFM In a steady state 2D plane session, when user want design an Induction Motor and display iron Losses versus Slip. Before solving, user must create an I/O physical parameter “SLIP”, controlled via a scenario. Next, the flux project must be solving with a multi parameter depending from “SLIP” /
3D Geometry	**ModifyCoordinateSystemForPoints .PFM** Modify coordinate system of a list of points (useful to add parameters after geometry import).
	SearchLinesMinimumLength.PFM Search lines with length lower than a given value.
3D Mesh	**FindOutNodesInVolumes.PFM** Display the number of nodes in selected volumes.
3D Physics	**ExtrudeFaceWithVolumeRegion.PFM** Extrude faces, and assign the new volumes to the volume regions, situated at the other side of the selected faces.
	ExtrudeFaceAndAssignSameVolumeRegion.PFM Extrude faces, and assign the new volume to a specific volume regions.
3D Postprocessing	**CreateSensorFor3DSlotForce.PFM** Create sensors to compute force on slots for motor (3D or skew).

Continued on next page

Macros (suite)

Exploitation SKEW	**BertottiIronLossesVsSlipAcImSk.PFM** In a steady state Skew session, when user want design an Induction Motor and display iron Losses versus Slip. Before solving, user must create an I/O physical parameter “SLIP”, controlled via a scenario. Next, the flux project must be solving with a multi parameter depending from “SLIP”.
General	**CreateIOTabulatedParameterFrom2DCurve.PFM** Create a new tabulated I/O parameter from a 2D curve
	BHSplineLaminationEquivalentCurve.PFM Create the new equivalent laminated material with the equivalent B(H) curve if we only have the B(H) curve of the non-laminated material.
	ExtractFrom3DCurve.PFM Extract specific values from 3D curve and create a new 2D curve. For instance extract rms values versus parameter 1 of all curves versus parameter 2.

For more details about all macros, please consult the document on the following path: C:\Cedrat\Extensions\Macros\macros_list_EN.doc

1.3. V11.2 new features: Flux documentation

Introduction

This page permits to give **some reminders** about the operation of Flux documentation and also to integrate some V11.2 new features:

- **new type of document** called **Best practices** (since the V11.1)
- **user portal** (since the V10.4)
- **new tools :** Excel technical sheet (V11.2 new features)
- **Easy access to 2D and Skew examples** by the supervisor (V11.2 new features)

Document supports

It exist two document supports:

- « **PDF** » documentation installed in local ans also available on the user portal
- « **HTML** » documentation accessible from :
  - Flux software (in the menu **Help,** point on **Help**)
  - Supervisor (click on the icon and on **Help**)

User Guide (PDF)

For each new version of Flux, a «PDF» user guide is associated.
It is automatically installedat the installation of Flux in :
C:\Cedrat\FluxDocExamples_11.2\UserGuide

The user guide contains :

- 4 volumes
- The new features document

The various PDF documents of user guide are accessible from the supervisor (click on the icon , and on **Documents**)

User portal (New)

The user portal is a sharing platform on which customers access to different services as the consultation of majority of « PDF » documentation.
The advantage of this platform is that the user may find updated documents or new documents between released versions.

Access to user portal : https://gate.cedrat.com

Best practices

A **new document type** is delivered since the 11.1 version called **Best Practices**. The goal is to bring advice on specific topics.
The existing documents are presented below.

"Best practices" documents	Language	
ModelisationDesToles.pdf	FR	Help to model magnetized strip.
ModelingOfMagneticCore.pdf	EN	
MaillerlEeffetDePeau.pdf	FR	Help to mesh the skin effect with a macro
MeshSkinEffect.pdf	EN	
ImportCAO.pdf	FR	Help to use CAD import in Flux.
CAD_Import.pdf	EN	
ReglerlLeMaillageLocalement.pdf	FR	Help to use Aided mesh with adjustment the mesh locally.
AdjustMeshlocally.pdf	EN	

New Tools

These new tools are Excel technical sheets (with PDF documents) which will allow for the user to:

- Determine the coefficients of Bertotti model of losses
- Determine the coefficient of analytic model of B(H) with 2 or 3 coefficients

Version	Folder	Documents
11.2 Dec 2013	BertottiLosses Coefficients	• BertottiLossesCoefficients.xls • DetermineBertottiCoefficientsForIronLossesComputation_EN.pdf • iron_losses_bertotti_model_fiorillo.pdf
11.2 Dec 2013	FitMaterialAnalyticModelsWithExperimental Data	• determine_coefficients_for_analytic_models.xls • FitMaterialAnalyticModelsWithExperimentalData_V01_E.pdf

Easy access to examples

With the aim to highlight and facilitate access to the examples provided with Flux, a context of the supervisor has been implemented (**Open example**).
The user has access via this context:

- the summary sheet (French and English) of the selected example
- the full pdf document (Only English) of the selected tutorial
- the opening of Flux project of the selected example in the state of execution requested (model with mesh, model with physics, solved model, post-processed model)

For more details on the **Open example** context, consult the map :
2.1.2 Contexts of the Supervisor.

Continued on next page

Resuming table Here is a summarized table of different type of documents as well as the consultation possibilities.

Document type	**PDF**				**HTML**
	Hard disk user	**User portal**	**Supervisor (documents)**	**Supervisor (Open example context) NEW !**	**Flux (Help) or Supervisor (? +Help)**
User guide (principles) (FR and EN)	✓	✓	✓	✗	✓
Installation guide (FR and EN)	✓	✓	✓	✗	✓
Example 2D (EN)	✓	✓	✓	✓	✗
Example 3D (EN)	✓	✓	✓	✗	✗
Example Skew (EN)	✓	✓	✓	✓	✗
Best practice (FR and EN)	✓	✓	✗	✗	✗
Tolls (Excel technical sheets) (EN) **NEW !**	✓	✗	✗	✗	✗
User guide (software aspects) (FR and EN)	✗	✗	✗	✗	✓

Reading advice For more information about the documents and examples, please consult the file **DocExamples_Readme.pdf** at the following path : *C:\Cedrat\FluxDocExamples_11.2*

2. V11.2 new features regarding Environment-Ergonomics

Introduction

This chapter deals with the V11.2 new features regarding the environment and the ergonomics.

Contents

This chapter contains following topics:

2.1. New supervisor

Introduction

In 11.2 version, the supervisor ergonomy has been modified fully. In particular, this new supervisor allows facilitating:

- the Flux access
- the examples access
- the edition and execution of PyFlux command files.

This section describes the **Flux Supervisor**, with which you can run Flux modules and manage your Flux project files and directories.

Contents

This section contains the following topics :

2.1.1. Environment of Flux Supervisor

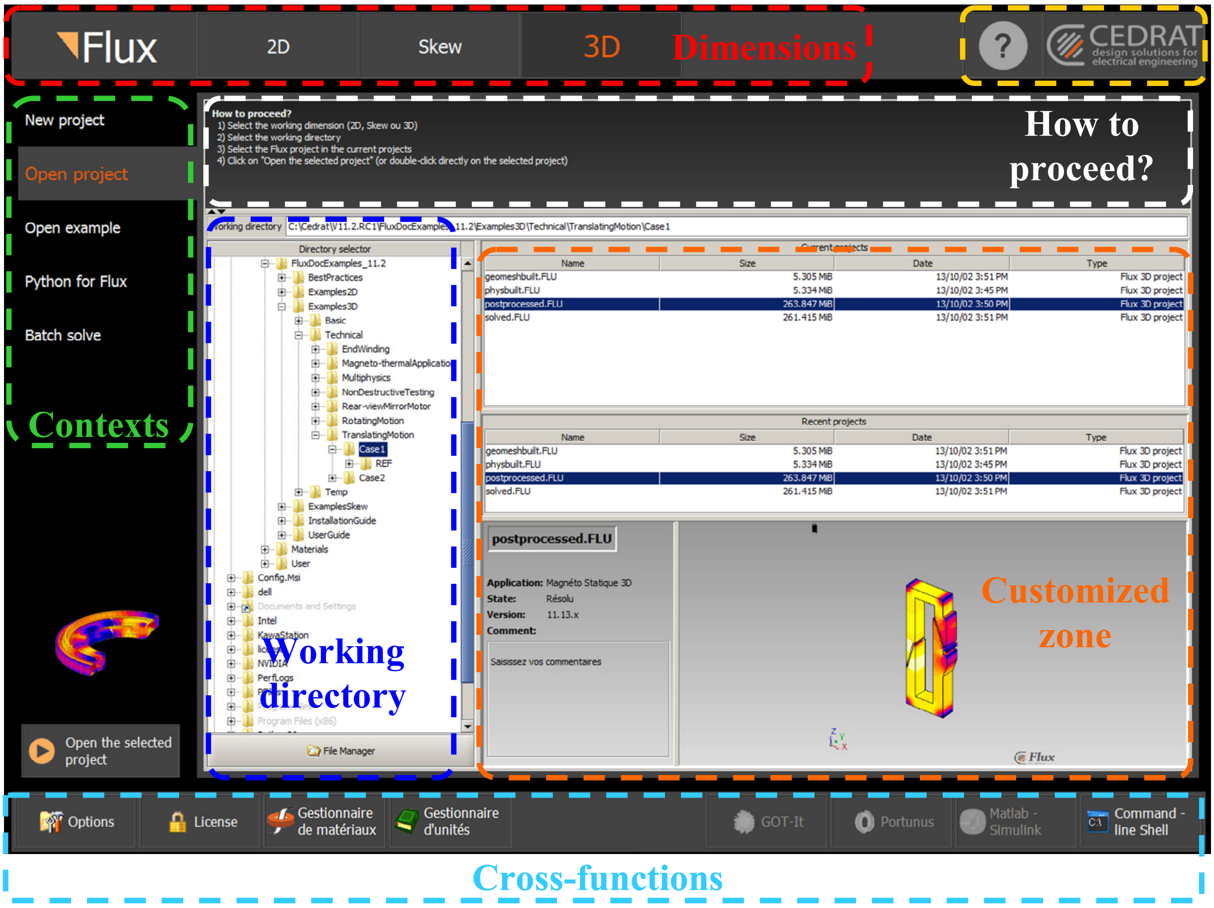
Access

To start the Flux Supervisor from the Windows taskbar, proceed as follows:

- point on **Start/ All programs/ Cedrat** (or your installation directory) and click on **Flux**

The Supervisor Window

The Flux Supervisor window is divided into several zones. The different zones are identified in the figure below and then detailed in following blocks.



Continued on next page

Zones of the Supervisor

The different zones of the Flux Supervisor and their functions are presented in the table below.

Zone	Function
Dimensions	The user selects the dimension in which he wishes to model his project: 2D or 3D, Skew
Contexts	The user have the choice between several use contexts of supervisor: • New project • Open un project • Open example • Python for Flux • Batch solve
Working directory **Directory selector**	The user chooses a working directory. The path of this directory is displayed. It is possible to manage folders and files by clicking on button : 
Customized zone	The content of this zone is adapted according to the context of use chosen. The action button  is also customized.
How to proceed ?	The process of use of each context is in this zone. It is possible to hide/display this zone by clicking on
Cross functions	The user also has access by the supervisor at cross-functions: • Specific functions to Flux (Options, License, Materials, Units) • Functions of coupling with external softwares (Got-It, Portunus, Simulink ...)
	This icon allows to access to : • Help (HTML documentation) • PDF documents (user guide, tutorials, new features document…) • User portal (sharing plateform) • …

2.1.2. Contexts of the Supervisor

Introduction

The supervisor contains several contexts of use:

- New project
- Open un project
- Open example
- Python for Flux
- Batch solve

To change the context, just click on the desired context.

New Project context

This context allows the user to **directly open the Flux software** with a **new project** in the **selected dimension**. The project is created in the working directory selected.

To start a new project, click on:

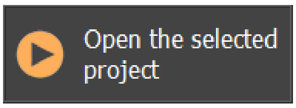
Open project context

This context allows the user to d'**open an existing Flux project.**

The customized zone contains several subzones:

- List of **current projects** contained in the working directory selected
- List of **recently opened projects**
- **Graphic view**: preview of model associated with the *.FLU project selected
- **Informations** on selected *.FLU project: Application, State, Comment entered by the user (to enter in this zone), …

To open an existing selected project, clicking on:
*(or double click on the *.FLU project)*



Open example context

This context allows to **open one of the proposed example** in the same required state (model with mesh, with physics, solved, post-processed).

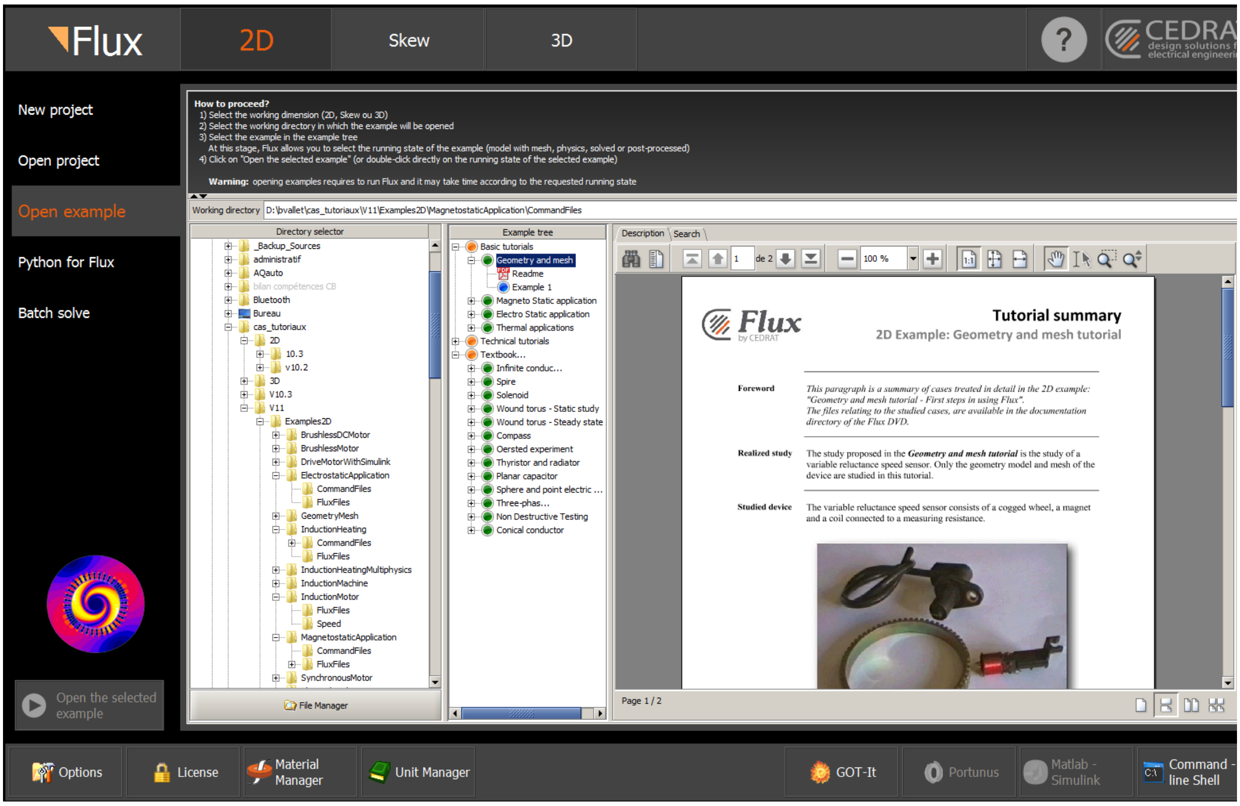
The opened example is **automatically saved** in the working directory selected.

La zone personnalisée contient plusieurs sous-zones:

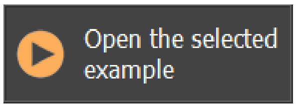
- **Example tree** with an tree structure with several levels:
  - **Group** (basics, technicals, textbook cases)
  - Studied **Tutorial**
  - **Example** (several examples are available in a same model)
  - **Running state**
- Viewing **the summary of the selected tutorial**
- **Search** function that allows filtering the examples in the tree structure

It is also possible to open the **PDF document** of the full tutorial by clicking on **Readme**.

Continued on next page



To open an example in the running state selected, clik on:
(or double click on the running state)



A **search** function (Tab **Search**) is available which allows filtering by keywords of the list of examples in the **example tree**.

Python for Flux context

This context allows manipulating python scripts contained in the selected working directory:

- Create files *.py
- Edit files *.py
- Execute files *.py

The customized zone contains several subzones:

- **List of python scripts** contained in the selected working directory
- **Python Editor**

Batch Solve context

This context allows to solve in batch mode (allows for example to reduce the computed time for complex models)

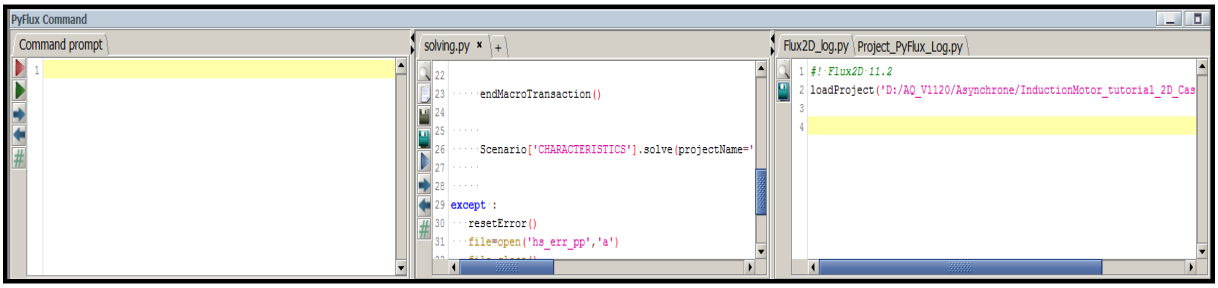
It is also possible to plan several resolutions of *.FLU projects(ready to solve) contained in the selected working directory.

2.2. Python editor

Introduction

A new python editor is now available in Flux.
The old "**PyFlux command**" window is divided in three parts :

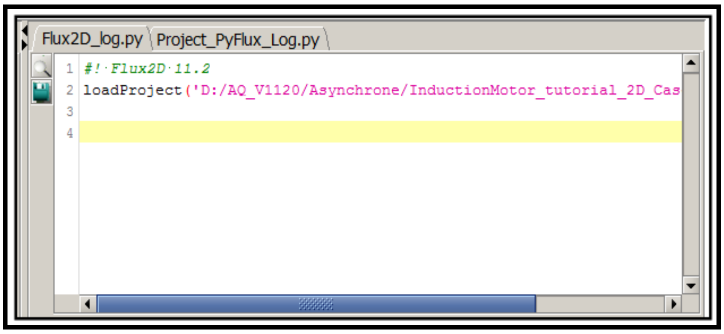
- On the extreme right, are **log .py files** to save command performed by the user during a flux session or flux project.
- On the middle, an **editor python** zone to open, read and create script.
- On the extreme left, a **"Command Prompt"** zone.

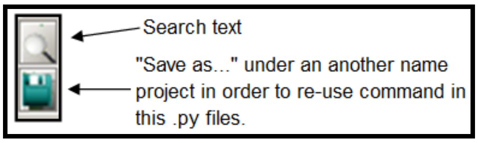


The .py flux log files.

In the pyFlux command zone, at the extreme right windows, two log.py files is avalaible (in read only) :

- the **"Flux[XX]_log.py"** : all commands performed by the user during a session are recorded in this file.
- the **"Project_PyFlux_log.py"**: all commands performed by the user during the beginning of the project up to the end of the project are recorded in this file.

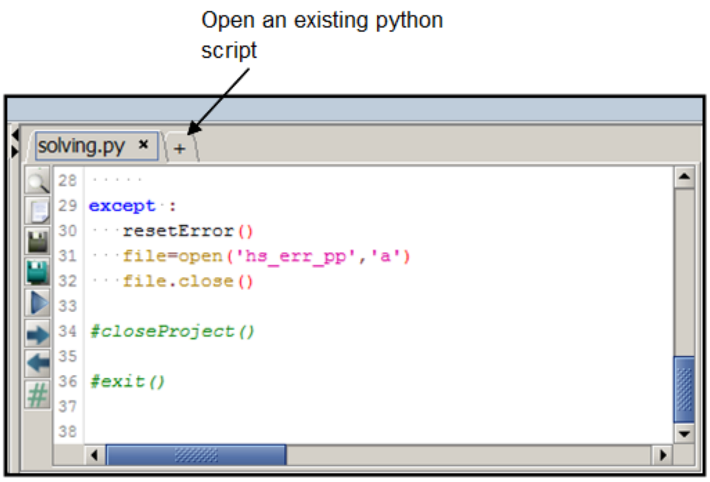


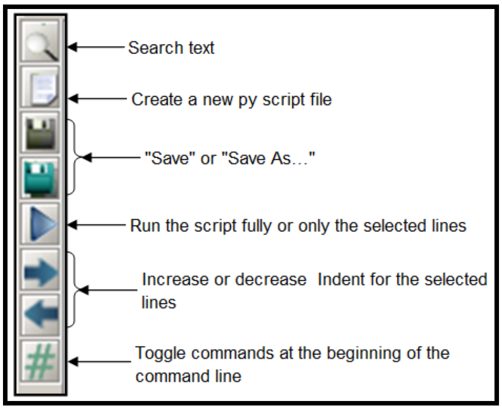


Continued on next page

The central zone : "PyFlux Editor".

With the central zone, the PyFlux editor, user can **read**, **write**, **run** totally or only some selected command of the python script.



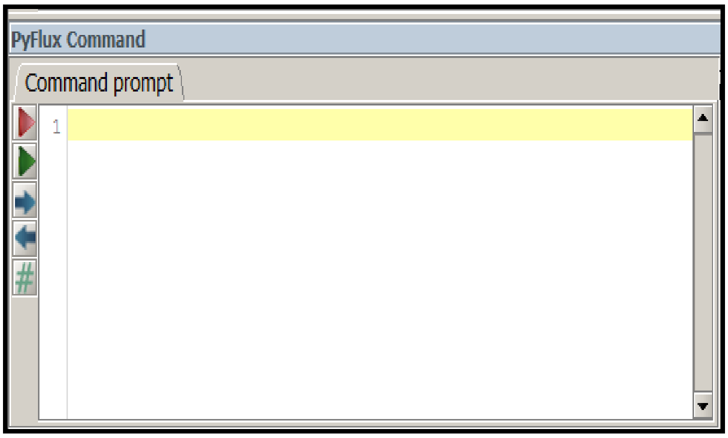


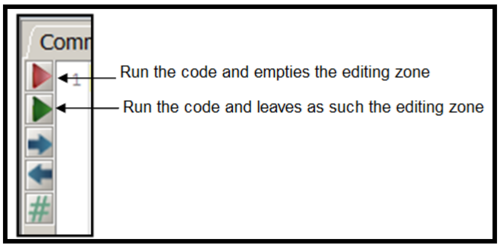
Notes :

- It is possible to open several py script file (one tab by file)
- This zone allows displaying automatically the groovy file when the user use subroutine in groovy language
- An editor python similar is available directly on the supervisor (script python context)

Command prompt" zone

With the "command prompt" zone, the user can launch and test a python command (or more python commands) directly, without create a new .py file. It's an operating direct mode, useful when you need to work quickly or debug your python command.

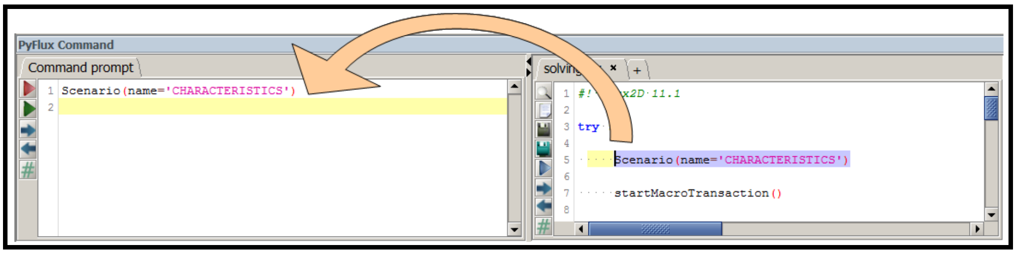




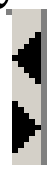
Continued on next page

Drag and drop

User can “Drag and drop” between different windows by selecting command with the mouse. It is equivalent to a copy + paste



Hide/display

User can hide or display one of these three zones in the “**PyFlux command**” window by clicking on 

2.3. Windows layout

Introduction

It is possible to modify the look of the Flux window on the screen, i.e.:

- modify the background color
- display / hide certain zones
- resize (reduce / enlarge) zones

Modify the background color

To modify the background color (reverse video):

- in the View menu, click on Reverse video

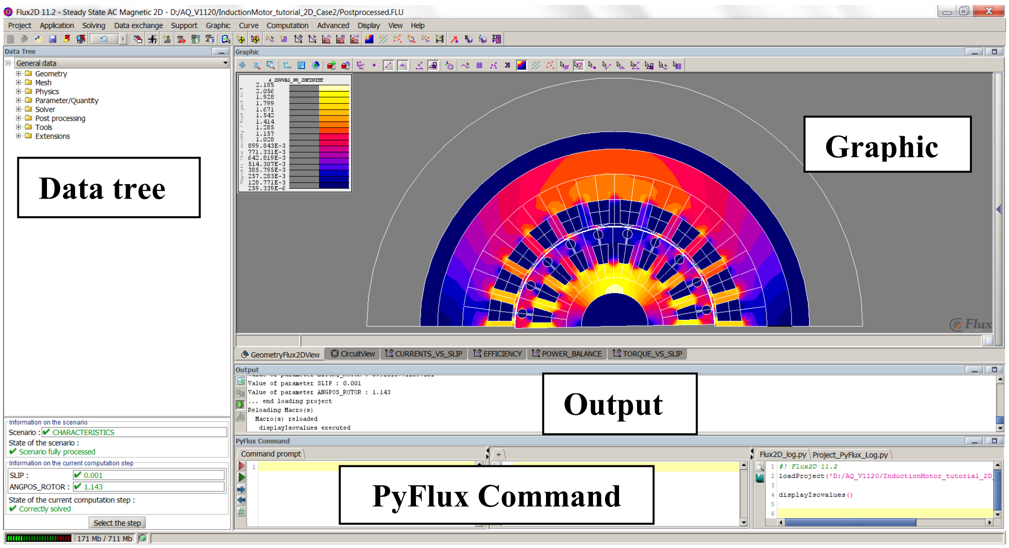
Display / hide zones

Use the icons located on the right of the "Title" bar of the window.

Windows

The Flux environment contains several windows :

- Data tree : contains all data of your project
- Graphic : contains the representation of the studied device with the graphic result (isovalues, curve, arrows …)
- Output : contains the result of each action executed
- PyFlux Command : contains the new editor python and the memory of actions achieved in the project



Continued on next page

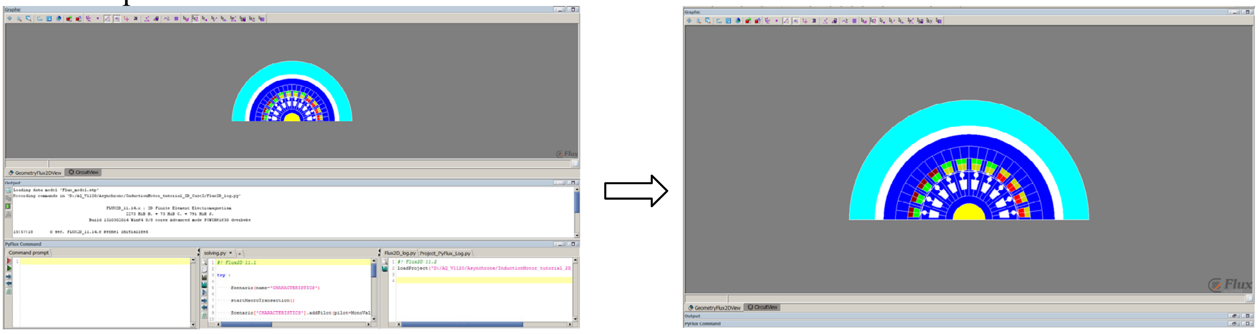
Minimize a window

To minimize (or hide) a window :
- click on 

or
- double click on the blue barre

The button  is transformed in  and the other windows are automatically resized

In this example to display only the graphic view, the user must minimize :
- the pyFlux command window
- the Output window



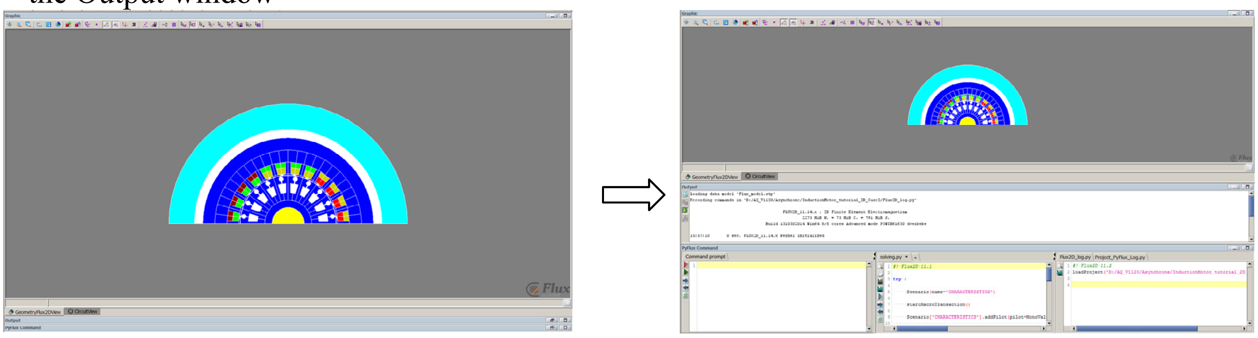
Resize a minimized window

To Resize a minimized window :
- click on 

or
- double click on the blue barre

The button  is transformed in  and the other windows are automatically resized

In this example to display the three window, the user must resized the minimized windows :
- the pyFlux command window
- the Output window



Continued on next page

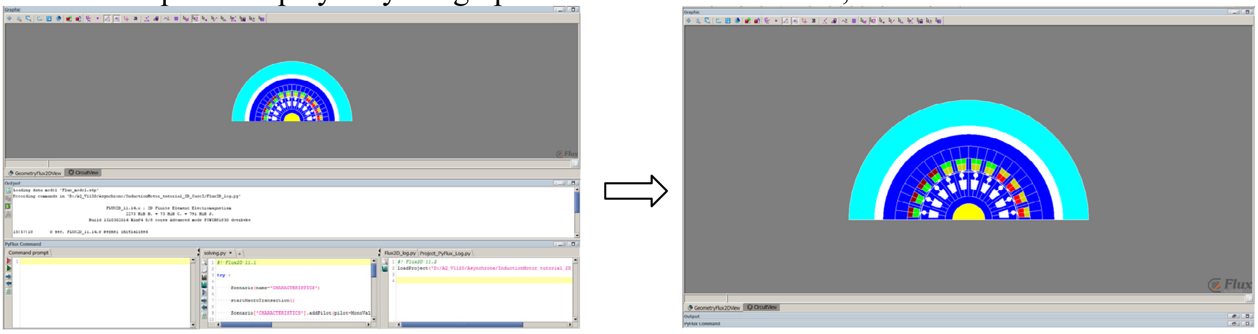
Maximize a window

To maximize a window :

- click on

The button is transformed in and the other windows are automatically minimized.

In this example to display only the graphic view in the full screen, the user must click on



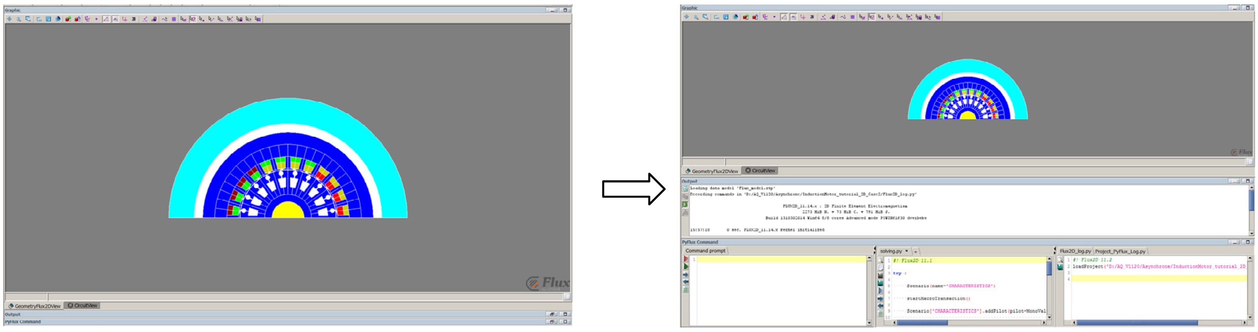
Reduce a maximized window

To reduce a maximized window :

- click on

The button is transformed in and the other windows are automatically resized.

In this example to display the three window, the user must reduce the maximized graphic view window.



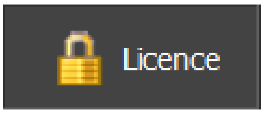
2.4. New features about license tools

Tools to manage the license

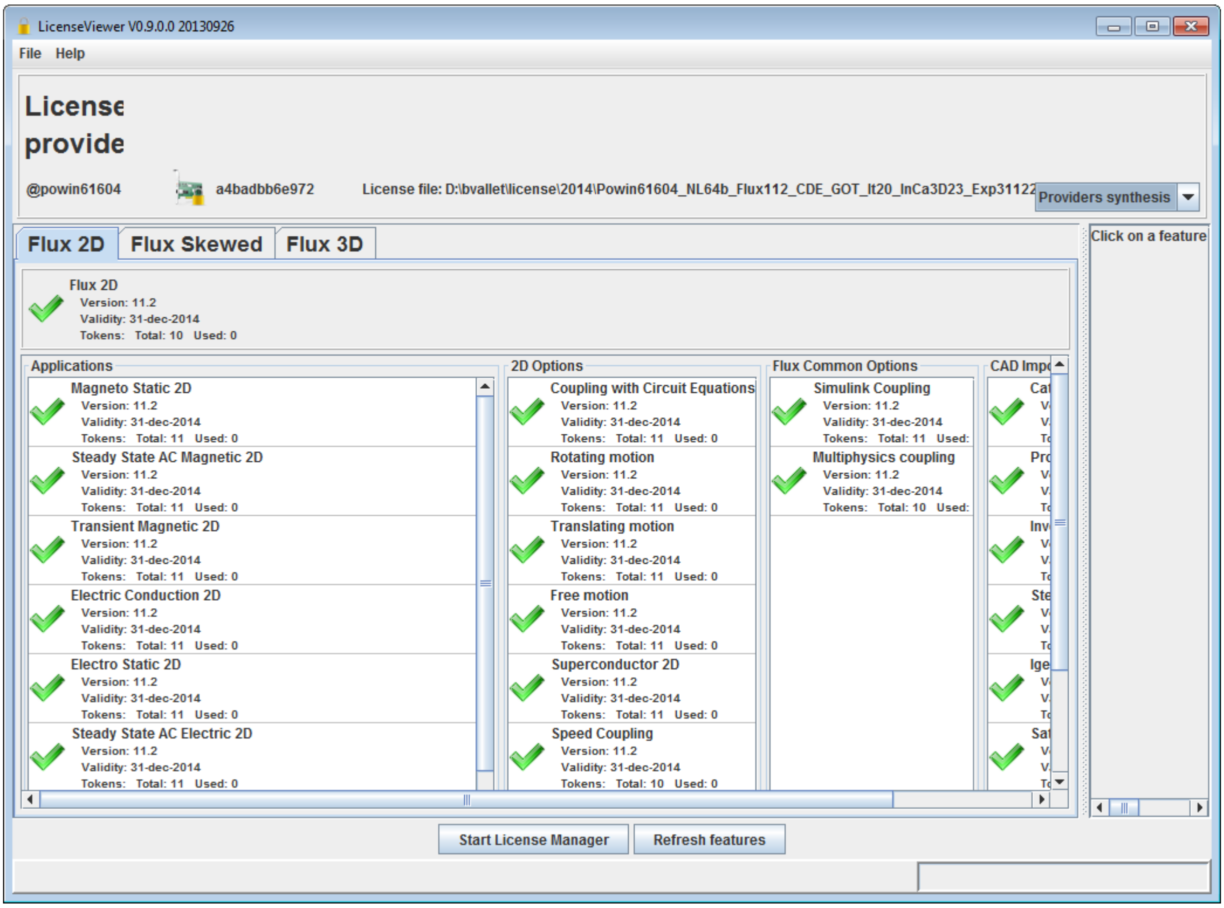
To manage the license, two tools are available for the user :

Tool	Description
The license viewer	The **license viewer** is used to display the authorized applications, authorized versions, the expiration date, the total number of tokens used ...
Le license manager	The **License manager** is used to configure the license server and connect it Flux to the license system.

Open the license viewer

To open the license viewer, click on: 

Le licence viewer is presented below.



Continued on next page

The possible states

The license is composed by several applications; the license viewer gives the state of each application. The possible states are:

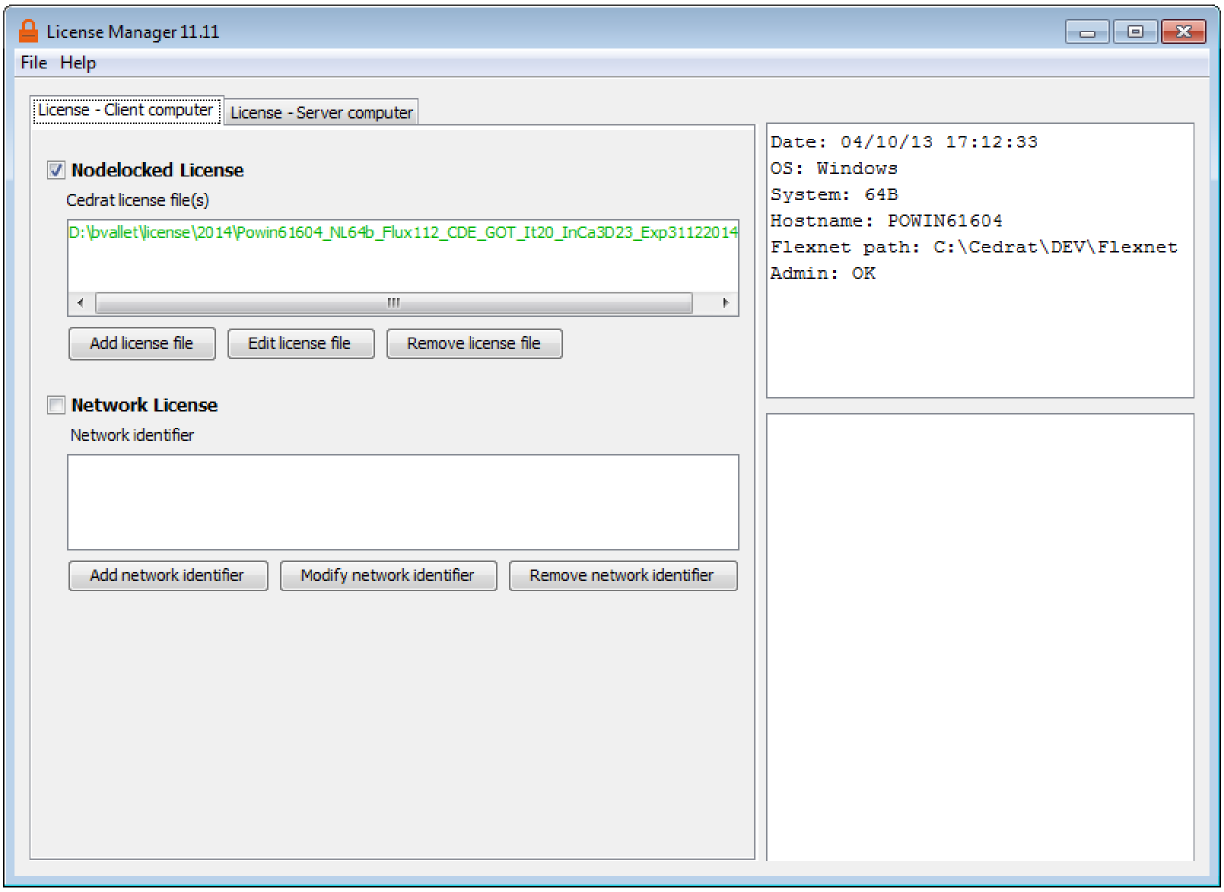
-  means that the application is included in the licence file and is valid, so available..
-  means that the application is included in the licence file but not valid, so not available.
-  means that the application is not included in the licence file so not available.

Open the license manager

To open the license manager :

- Start from the license viewer, click on the button **Start License Manager**

The license manager is presented below.



Reading advice

The user will find detailed information about license manager in the “Installation guide”.

3. V11.2 new features regarding Geometry

Introduction

This chapter deals with the V11.2 new features regarding the geometry.

Contents

This chapter contains the following topics:

3.1. New import formats

Introduction

FLUX software has the ability to communicate with other software packages and to carry out the transfer of data from CAD tools to the Finite Element (FE) analysis tools.
Different formats and different processes are available:
- import of geometry starting from geometric files
- import of geometry starting from mesh files
- import "Advanced Mode"

Intérêt

The import of a CAD geometry file into FLUX takes into account projects possessing **complex geometries** (e.g. presenting **twisted surfaces)**. These types of surfaces cannot be generated directly using the available tools in FLUX.

Nouveaux formats

The new formats are :
- SOLIDWORKS (extension :*.SLDASM, *.SLDPRT)
- PARASOLID (extension :*.PRT)
- NX (extension : *.X_T, *. XMT_TXT, *.X_B, *.XMT_BIN)

These formats are available only for the import, and not for the export.
These formats are not available with Linux OS.
These formats are available for the import in « advanced mode ».

Continued on next page

All import formats

Here is a table of all import formats available.

Type of import	Available File formats	Extension	Type of format
geometry import called « standard »	IGES (Initial Graphics Exchange Specification)	*.IGES, *.IGS	standard
	STEP (Standard for Exchange of Product)	*.STEP, *.STP	
	DXF (Draw eXchange File)	*.DXF	
	STL (STereo Lithography)	*.STL	
	FBD (Flux 2D géométrie)	*.FBD	proprietary
mesh import	NASTRAN neutral	*.NAS, *.DAT	standard
	PATRAN neutral	*.PAN, *.DAT	
	UNV (UNiVersel Ideas Master Serie)	*.UNV	
	MED (Model of data exchange)	*.MED	
geometry import called « advanced mode »	IGES (Initial Graphics Exchange Specification)	*.IGES, *.IGS	Standard
	STEP (Standard for Exchange of Product)	*.STEP, *.STP	
	SAT	*.SAT	proprietary
	CATIA V4	*.MODEL	
	CATIA V5	*.CATPRODUCT *.CATPART	
	INVENTOR	*.IPT	
	PROE (Pro Engineer)	*.ASM , *.PRT	
	SOLIDWORKS (New !)	***. SLDASM , *.SLDPART**	
	NX (Siemens) (New !)	***.X_T, *. XMT_TXT, *.X_B, *.XMT_BIN**	
	PARASOLID (New !)	***.PRT**	

3.2. Improvements and new features about Sketcher

Introduction

The sketcher is a « drawing » tool that facilitates easier planning and execution of CAD 2D geometries. The integration of a sketcher into Flux software carried out in the version V11.1 brings about an easier and better performing description of the geometry in order to be similar to the current, globally used CAD tools.
The version V11.2 of Flux consolidates the sketcher environment, bringing about improvements to existing functions, as well as adding new ones.

Contents

This section contains the following topics:

3.2.1. Graphical improvements

Introduction

The sketcher is a tool that has numerous graphical functions to facilitate the creation and manipulation of the geometry.
Various improvements have been made in Flux version V11.2.0 :

- Displacement of a selection by keyboard shortcut
- Choice of grip point during displacement
- Detection of the center of a line
- Displacement of a selection containing propagated entities
- Choice of options of the grid according to the user preferences (Supervisor)

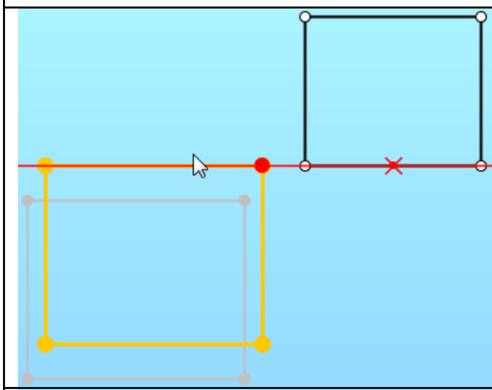
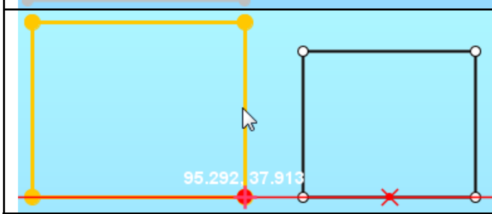
"Arrows" move

The user was already able to displace a selection graphically, using the mouse.
Now , the user can also carry out displacement of a selection by implementing the following keyboard shortcuts:

- Fine displacement (equivalent to the dimension of a subdivision of a grid cell) : **Ctrl + Arrows (keyboard)**
- Very fine displacement (equivalent to the distance between the magnetization points of the grid : **Shift + Ctrl + Arrows (keyboard)**

Grip point

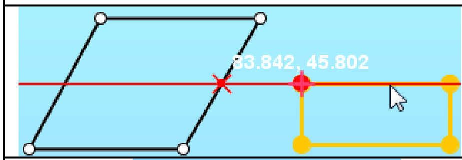
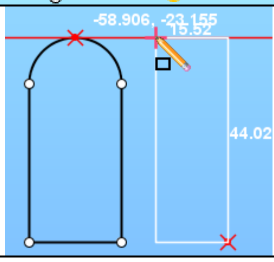
During the displacement of a selection, a point of the selection is identified as the grip point. This permits to detect straight lines of graphic support in order to align the selection with respect to other existing entities.
Until this version, the grip point was determined randomly in the selection and it was impossible to change it.
With V11.2.0 the user can change the displacement grip point by using the arrow keys (keyboard).

Examples	
	The operated selection comprises the 4 lines and the 3 points in bold yellow, as well as the grip point identified in red which permits the detection of the support straight line (here the horizontal straight line in red)
	By keyboard shortcut the user can choose one of the 3 other points as a grip point.

Continued on next page

Detection of the center of a line

During the creation and the displacement it is possible to detect the existing points in order to facilitate desired alignments.
With the V11.2.0, it is also possible to detect the center point of a line segment and the center point of an arc.

Examples	
	Detection of the center point of a line segment during a displacement
	Detection of the center point of an arc during a creation of a rectangle

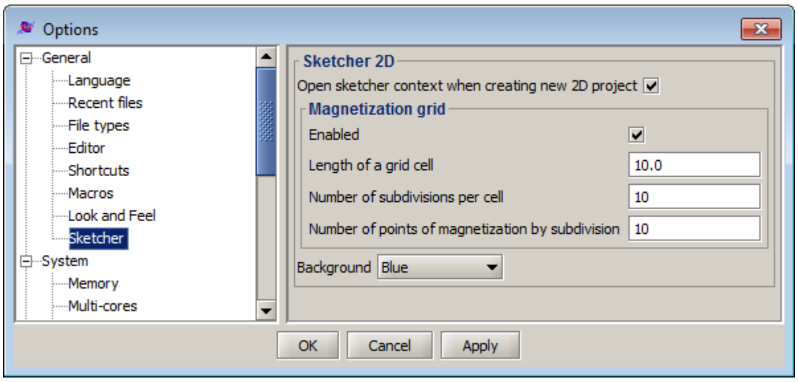
Displacing a selection with propagated entities

Globally, the propagated entities cannot be displaced directly. One must displace the entity of origin in order to be able to displace the propagated entity.
This rule has not been changed, but a particular case allowing displacement of propagated entities is now possible.
Let us suppose that the user operates a selection that is comprised of propagated entities and standard entities. If, in this selection there are the propagated entities as well as their standard entities of origin, then the displacement of the selection will be authorized.

This improvement will permit, for instance, the re-centering of geometry that comprises propagated entities and entities of origin.

Grid options

The grid can be parameterized in the sketcher options. This permits the user to choose the magnetization grid step for the creation and displacement of a selection. These options are specific to the current project.
With the version V11.2.0 the options of the grid has been added in the preferences in order to permit the user to choose the dimensions of the sketcher grid adapted to his ongoing projects. This option is accessible from the supervisor.



3.2.2. Improvements of management of propagated entities

Introduction

The propagated entities are connected to the entities of origin.
These entities are created after the construction operations (symmetry, repetitions ...) by ticking the option « Connected to the origin ».
With the version V11.1 the manipulations of the propagated entities were very limited; no correction or creation operations were possible. In V11.2.0 version this limitation has been eliminated in order to facilitate the manipulation of the propagated entities.

Improvements of intersections

The correction of the intersections now takes into consideration the intersections between:

- a propagated entity and a standard entity
- a propagated entity and another propagated entity.

The entity can be a point or a line.

Improvements of superimpositions

The correction of superimpositions takes into consideration the superimpositions between:

- a propagated entity and a standard entity
- a propagated entity and another propagated entity.

The entity can be a point or a line

Applied rule

The applied rule is simple:

- if there are intersections and superimpositions defects, the correction option transforms the propagated entities in conflict into standard entities.

Impact of the creation

There is an option of intelligent correction that automatically corrects the superimpositions which are being created.
With improvement to the management of propagated entities, the user can now create new entities starting from a propagated entity.

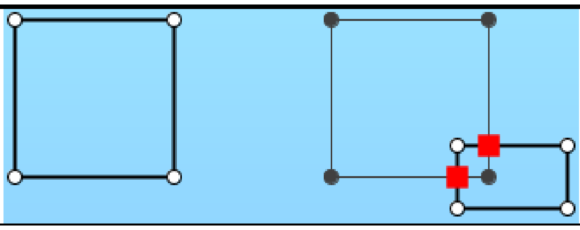
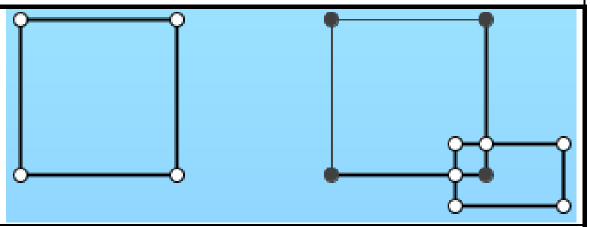
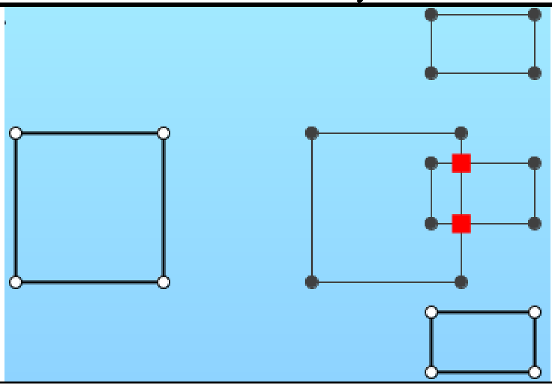
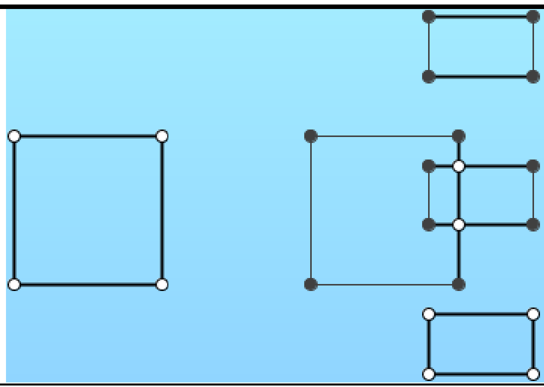
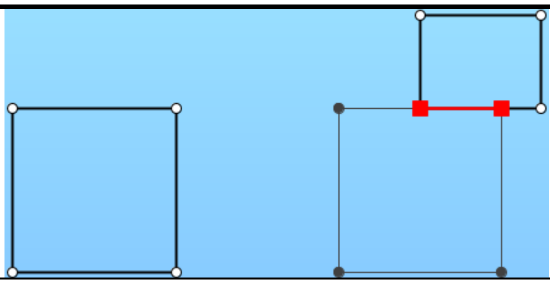
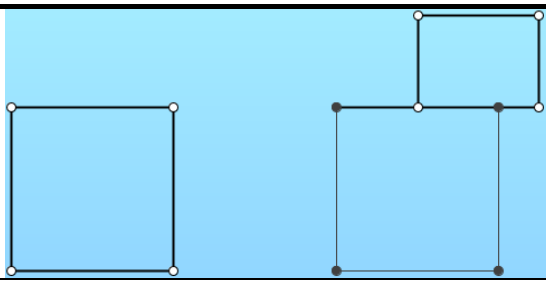
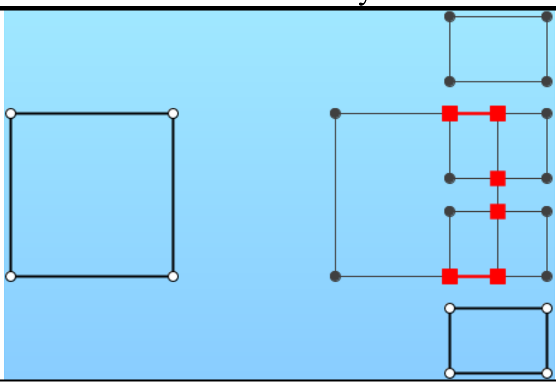
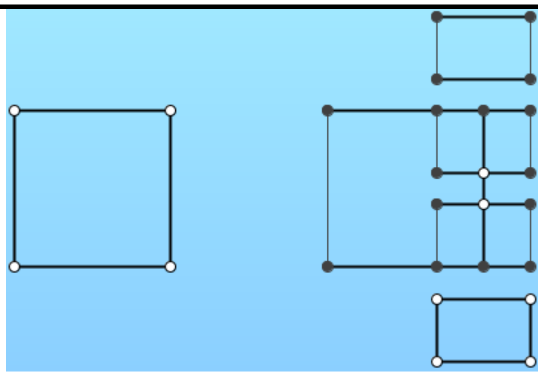
This automatic intelligent correction is also made for the displacement of a selection on a propagated entity.

Warning

The problems of intersection and superimposition of propagated entities with other entities normally originates with the user creating a geometric construction which is not correct.
Logically, if the user constructs the geometry accordingly, he should not encounter this type of defect. Nevertheless, this improvement will permit inexperienced users to unblock certain problematic situations.

Continued on next page

Example Here is few examples:

Before correction	After correction
	
2 intersections between propagated entity and standard entity	The propagated entities on conflict have been converted in standard entities
	
2 intersections between propagated entity and propagated entity	The propagated entities on conflict have been converted in standard entities. The conversion is reflected on linked entities
	
1 superposition between propagated entity and standard entity	The propagated entities on conflict have been converted in standard entities.
	
2 superimpositions between propagated entity and propagated entity + 2 intersections between propagated entity and propagated entity	The propagated entities on conflict have been converted in standard entities. The conversion is reflected on linked entities.

3.2.3. Improvement of management of a fillet

Introduction

With the version V11.1, the creation of a fillet uses an arc of **2 points without a center** point. Several rules of priority are set to manage the graphic displacement. These rules are not adapted to preserve a fillet arc, namely the segments tangent to the arc.
With the version V11.2, a type has been added to the creation of the fillet arc, permitting the user to:

- identify the arc as being a fillet
- preserve the fillet after certain displacements (segments tangent to the arc)
- preserve the fillet after the modification of the radius of the arc (segment tangent to the arc)

Fillet identification

After the construction of a fillet, a graphic symbol (**F** letter) is added next to the arc created.

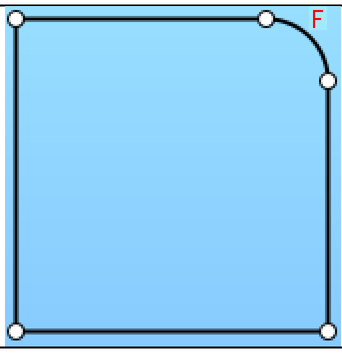
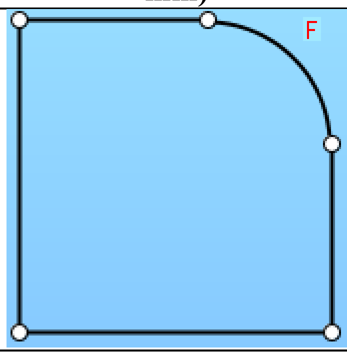
Move

With this improvement, the arc type fillet is preserved during the move of :

- the fillet arc
- a segment tangent to the fillet arc
- an extremity point of the fillet arc
- the assembly arc + its 2 tangent segments

Modify the radius

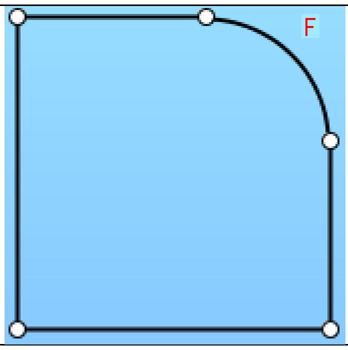
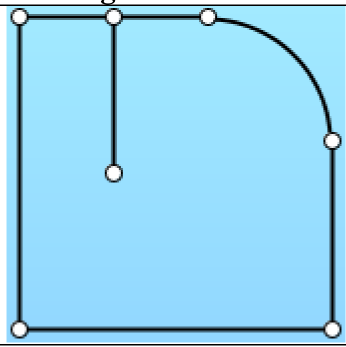
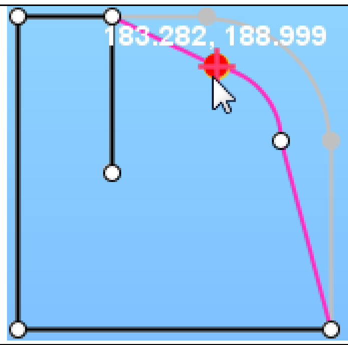
With this improvement, the modification of the radius of the fillet arc permits preservation of the fillet.

Initial fillet with a radius of 5 mm	**Modification of the radius (10 mm)**
	

Continued on next page

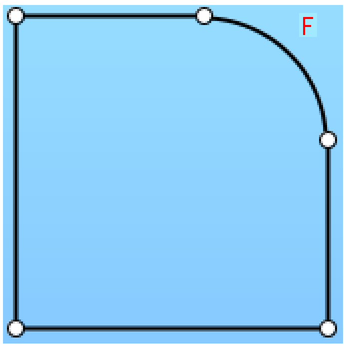
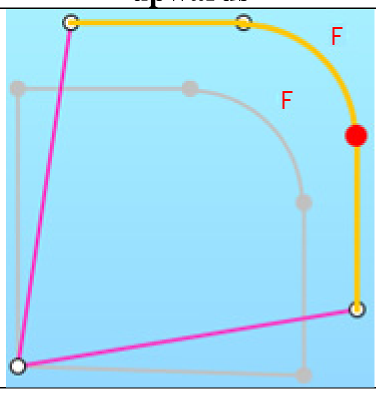
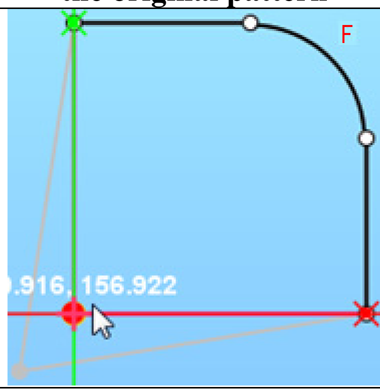
Actions on tangent segments

The operations carried out on the segments tangent to the fillet arc are authorized, but as a counterpart the type of the arc is lost. The arc is then no longer considered as a fillet, the symbol (**F**) has disappeared.

EXAMPLE		
Initial fillet	**Creation of a segment starting from a segment tangent to the arc**	**Move an extremity point of the arc**
		
The symbol (**F**) is displayed, the arc is a round type	The initial tangent segment is fractionated. The symbol (**F**) is disappeared	The type “fillet” is lost, the tangencies with the arc are not conserved

Limitation

The lengths of the tangent segments are always preserved, irrespective of the displacement carried out.
This is not a dysfunction, but a limitation which will be removed in a future version by integrating constraints into the sketcher.

EXAMPLE		
Initial fillet	**Displacement of fillet arc upwards**	**Displacing the point to obtain the original pattern**
		
Original pattern with angles at 90°	The original pattern has been modified	With a supplementary stage of displacement the original pattern is recovered

3.2.4. Improvement of the functionality "Measure"

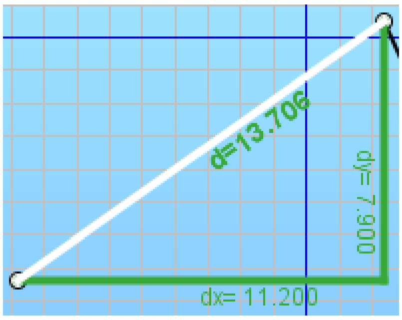
Introduction

In the sketcher it is now possible to measure :
- a distance between two points
- a line segment length
- a line arc length

The information is given graphically (V11.1) and also in the **Exit** window (new features with V11.2 : adding of python commands for each type of measure)

Point –point measure

The measure option for the distance between two existing points returns the following information:
D, Dx and Dy



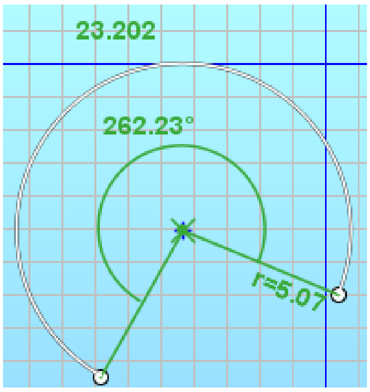
It is possible to consider a free point on the graph for the measure option. In this case, the information returned is only graphic, as the python command requires that a point should be a Flux geometry point.

Measure of a segment

The measure option for a segment is a graphic shortcut to obtain the measure point (point related to the two extremity points of the segment). Therefore, the same information is obtained D, Dx and Dy.

Measure of an arc

The measure option for an arc gives the following information :
Radius, Angle, curvilinear abscissa and the coordinates of the center point



The coordinates of the center point are not graphically displayed to prevent graphic crowding.

3.2.5. Creation of a rectangle by his center

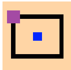
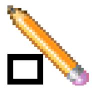
Introduction

It is possible to create a rectangle:
- by his diagonal (existing in V11.1)
- by his center (V11.2 new feature)

** The created rectangle is not an entity as a full part but only 4 points and 4 "segment" lines.*

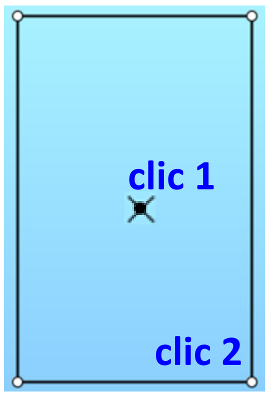
Access / Cursor

The different accesses and the personalized cursor for this mod of creation are presented on the following table:

Access		Cursor
icon		
menu	Construction → Rectangle → Rectangle center	

Creation Rectangle center

The creation process of a **Rectangle center** is presented in the table below.

Stage	Description	Illustration
1	Activation of the mode	
2	1st left click: fix the center of the rectangle (fictive point)	
3	Moving mouse : • give a view of the future rectangle • give information of the creation (coordinates of the next point, width and length)	
4	2nd left click: • fix the rectangle • create the corresponding entities « Point » and « Line »	
→	The mode is still activated and the user can enchain with another creation of rectangle by performing again In the same order the steps starting from step 2	
→	The right click permits the deactivation of the mode	

Information of the creation

During the creation of the rectangle, several information are available:
- Coordinates of the point to set (cursor position)
- Width and length

Preview of the rectangle with information displayed during the creation


3.2.6. Implementation of "curve" line (CAD import)

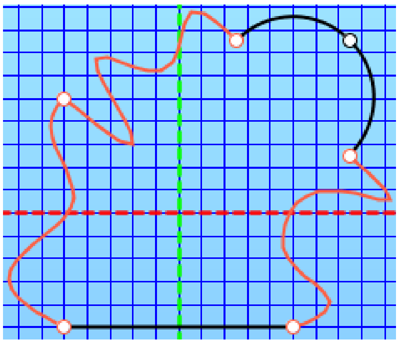
Introduction

In the context of the standard geometry of FLUX (without sketcher) it is possible to import a geometry that comprises lines called « curve » (also called « spline » or « complex line » or also « CAD line »).
With the version V11.1, these types of lines were not graphically interpreted in the sketcher context.
They were integrated in the sketcher context in this new version V11.2 with the ability to :

- Visualise curved lines
- Edit curved lines
- Correct the intersections
- Delete a curve line
- Create a curve line starting from an extremity point

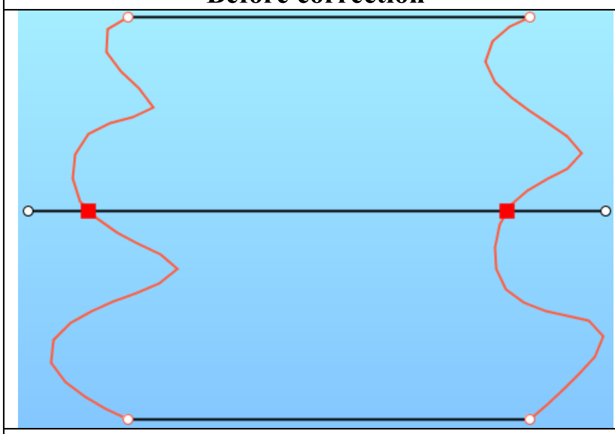
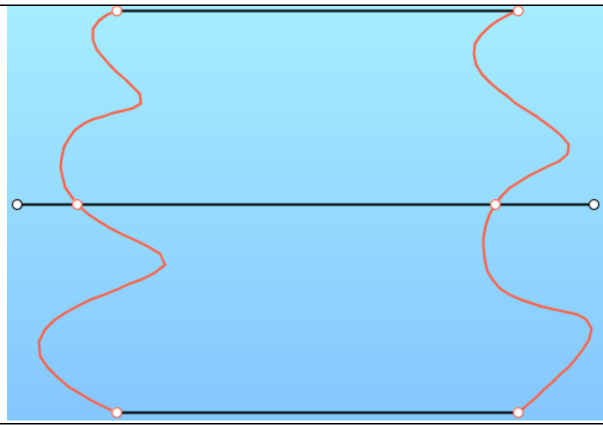
Display curve lines

The curve lines are automatically interpreted graphically.
They are identified by a red color to specify that they are not modifiable and not displaceable.



Correction of intersections

The global correction of intersections takes into account the intersections with curve lines. Here is an example below.

Before correction	After correction
	
2 curve lines with 2 defects of intersection	‘ curve lines and 0 defect

Continued on next page

Limitations

Several operations on the curve lines are limited:

- All the operations of local corrections (adjusting, dividing, extension of a line…) have no effect on the curve lines.
- The curve lines are not taken into consideration during the operations of construction (symmetry, repetitions …) with the option « connected to the origin » inactivated.
- It is not possible to copy/cut/paste a curve line.
- …

3.2.7. Import an image on the background

Introduction

The user has the ability to import an image in the background of the sketcher screen in order to get aid with creating of a geometry.
Then, using the background image on screen may be used to create the desired geometry.

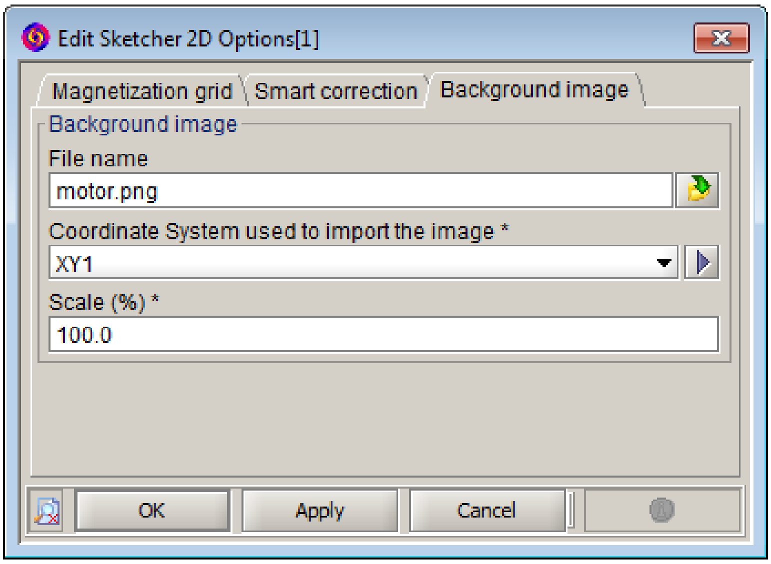
Image formats

The formats of the standard images are compatible with this function of image import. (png, jpg, …)

Import options

The import options of the image are as follows :

- Name of file
- Point of reference for importing of the image
- Resizing



Permanence of the imported image

The imported image is permanent. It is linked to the project during the saving.

Display / Hide image

There is a filter that permits the display or hiding of the charged image (menu **Display/view**, command **Display background image**)

Continued on next page

Import background image

To import an image to the background of the screen :

Step	Action
1	Open the options box : • In the menu **Options** click on **Edit**
→	The dialog box **Edit Options Sketcher 2D** opens
2	Activate the tab **Background image**
3	Choose the image to be imported
4	Choose the mark for centering of the image
5	Choose the sizing of the desired image
6	Validate by clicking on **OK**
→	The dialog closes, and the imported image will appear at the background on the graph

Modify an imported image

It is possible to modify the options of the imported image (coordinate system, dimension…) by editing the options box again.

It is possible to adjust height/width dimensions by clicking on : 

Graphic option

After the import of the image, the user can adjust the transparency of the image at the background, by moving the cursor of the **transparency bar**.

3.2.8. Management of geometric defects

Introduction

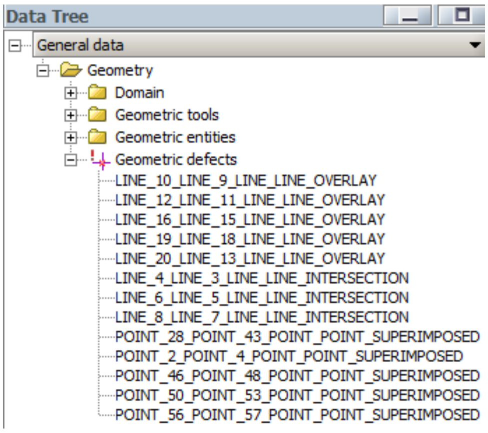
Within the context of the Flux standard geometry (apart from sketcher) it is possible to detect and display the geometric defects.
The management of defects has been added in the sketcher with the ability to :

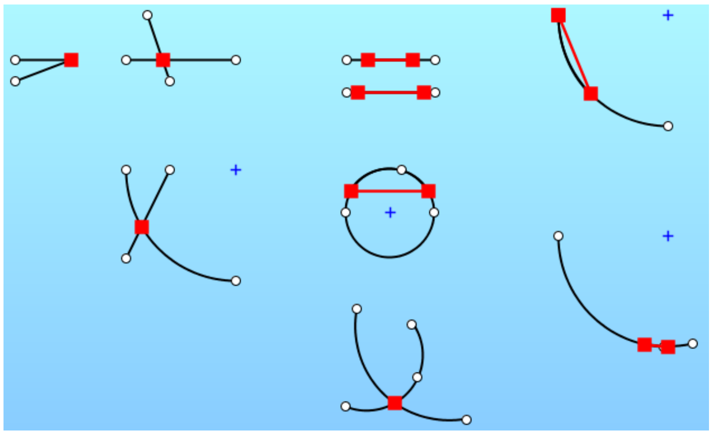
- Detect the defects (by the command "**Check geometry**")
- Display/Hide the defects
- Correct the defects (existing correction functions)

Detect defects

To detect the defects, the command "**Check geometry**" must be executed in the menu **Tools**.
The detected defects are automatically displayed and listed in the data tree.
The defects taken into consideration are :

- Intersection between entities (point/line or line/line)
- Superimposition between entities (point/point or line/line)





Display / Hide defects

There is a filter allowing to display or hide the defects (menu **Display/View**, command **Display geometric defects**)

Heal geometric defects

To correct all the detected defects there are three possible commands :

- Correct all the intersections
- Correct all the superposition
- Correct all the geometry (intersections + superposition)

To correct one defect specifically, the local correction commands must be used :

- Defect of intersection : adjust the intersections, divide
- Defect of superposition: merge 2 lines.

3.2.9. Evaluate the parameterized formulas

Introduction

Parameterized geometry cannot be graphically manipulated in the sketcher. The parameters are integrated in the formulations of the geometry.

In particular cases of use, the user may wish to manipulate the geometry at will without having the constraints connected with a parameterized geometry. A new function has been added allowing the user to evaluate all the parameterized formulations.

Evaluate the parameterized formulas

The command **Evaluate the parameterized formulas** permits one to replace all the formulas depending on parameters by numerical values.
Then the user can manipulate the geometry at will.

Access

This command is accessible on the **menu Tools** or by the **contextual menu** obtained by right click on a graphic selection of entities.

Global/local application

To apply this command it is necessary to first select the entities.
The user can then apply this command on the assembly of the geometry, (selecting everything) or on one part of the geometry (selecting only the desired entities).

3.2.10. Convert the propagated entities to standard entities

Introduction

The user may wish to « break » the links between entities, obtained by propagation.
A new function has been added which converts the propagated entities into standard entities.

Convert propagated entities

The command "**Convert propagated entities"** converts propagated entities into standard entities.
The user can then manipulate the converted entities independently from one another.

Access

This command is accessible in the menu **Tools** or by the **contextual menu** and is accessed by a right click on the graphic selection of entities.

Global/local Application

To apply this command, one must first make a graphic selection of the entities. The user can then apply this command on the assembly of the geometry. Selecting all or a part of the geometry (he selects only the desired entities).

4. V11.2 new features regarding Physics

Introduction

This chapter deals with the V11.2 new features regarding the physics.

Contents

This chapter contains the following topics:

4.1. Modelling the pseudo-anisotropy (lamination) and crystalline anisotropy

Introduction

The anisotropic materials have a behavior law which depends on the field direction. Anisotropies can be of different natures :

- The crystalline anisotropy, connected with the crystalline structure of the material which originates from the manufacturing procedure
- the pseudo-anisotropy connected to lamination and modelled by the homogenization of the package of stacked sheet metal

This section presents new information regarding Flux software that takes into consideration these two forms of anisotropy :

- a new type of region (**Laminated magnetic non conducting region)** to model the pseudo-anisotropy connected with lamination
- new material models to model the crystalline anisotropy in 2D

Contents

This section contains the following topics:

Reading advice

For complementary information see the following documents :

- *Document of good practice « **The modelling of stacked sheet metal in Flux** » (updated with Flux V11.2)*
- *User's Guide, volume 2, chapter 1, « The materials : principle »*

4.1.1. General context / theoretical reminders

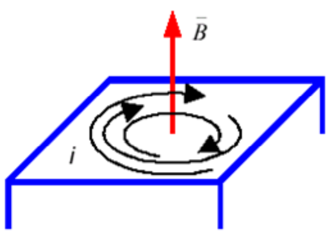
Introduction

This section presents several elements that enable the user to understand the working context. A more complete vision is available in the document of good practice : « ***The modelling of metal sheets in Flux*** »

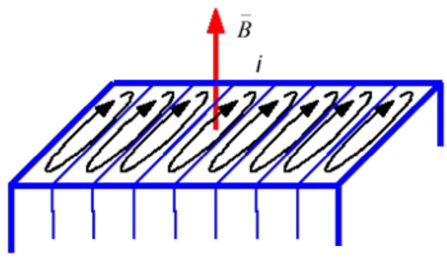
Magnetic core made of laminations

The magnetic cores of electrical machines and of transformers are in general made of laminated materials (stacked magnetic sheets) in view of limiting the eddy current losses.

The massive magnetic core is replaced by a stack of magnetic sheets insulated from one another. The purpose of the insulation is to prevent the circulation of the currents from a sheet to the other.



Massive magnetic circuit



Laminated magnetic circuit

Soft materials

The soft materials generally used have an **anisotropic, nonlinear, hysteretic** behaviour, and the corresponding material properties can also depend more or less strongly on other physical quantities, for example on **temperature** or **frequency**.

Accurate modelling of the magnetic sheet anisotropy remains difficult.

It is indeed possible to distinguish two forms of anisotropy:

- crystalline anisotropy due to the crystalline structure of the material
- the anisotropy associated with the heterogeneity of the structure (stacking up of magnetic sheets, composite material ...) ; therefore, we speak about «pseudo-anisotropy».

In function of objectives, the treatment of these two types of anisotropy can be quite different.

Pseudo-anisotropy

As regards the first form of anisotropy, **the pseudo-anisotropy**, it can be treated by homogenization methods meant to simplify the laminated structures and thus to compute the fields whilst ensuring reasonable computation time.

Continued on next page

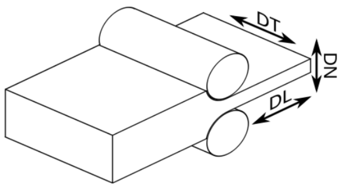
Crystalline anisotropy (in the sheet plane)

The models that permit the taking into consideration of the **crystalline anisotropy** are those models focused on the 2D anisotropy (i.e. only **in the sheet plane**).

These models mainly observe the measured magnetic characteristics of the sheets in their main directions, the direction of lamination **DL,** and the transversal direction **DT**.

The following models can be cited from the Nahil HIHAT thesis:
- separation of the axes model [NIYS75]
- model of the two axes [HLN84]
- elliptical model and elliptical model with axes rotation [DNP83]
- empirical anisotropic models …

Preferred directions of a sheet:
- DL: direction of lamination
- DT: transversal direction
- DN: normal direction



Bibliography

Complementary information on the modelling of anisotropic magnetic materials is available in the following documents:

- *"Quasi 3D Models for the Analysis of Structures Presenting a 3D Anisotropy"- thesis of Nabil HIHAT – 2010 - Université Lille Nord de France – UArtois*
- *"Anisotropic and nonlinear laws of magnetization: modelling and experimental validation" - thesis of Thierry PERA – 1994 - INPG ()*
- *"Contribution to the bi-dimensional and three-dimensional modelling of anisotropy phenomena in 3-phase transformers" thesis of Jean Marc DEDULLE – 1990 - INPG ()*

References regarding semi-analytical 2D models (plane of the sheet) :

- [NIYS75] T. Nakata, Y. Ishihara, K. Yamada et A. Sasano :
  Non-linear analysis of rotating flux in the t-joint of a three-phase, three-limbed transformer core.
  In proceedings of Soft Magnetic Materials 2 Conference, pages 57-62. 1975.
- [HLN84] D. Huttenloher, H.W. Lorenzen et D. Nusheler :
  Investigation of the importance of the anisotropy of cold rolled electrical steel sheet. *IEEE Transactions on Magnetics*, 20(5):1968 -1970, 1984.
- [DNP83] A. Di Napoli et R. Paggi :
  A model of anisotropic grain-oriented steel.
  IEEE Transactions on Magnetics, 19(4):1557-1561, 1983.

4.1.2. Modelling the pseudo-anisotropy: new type of region (1)

Introduction

This section presents the new region, **Laminated magnetic non conducting region,** and it gives the elements for basic understanding.

The issue

The use of the finite elements method requires dividing the modelled geometry into elementary meshes, while observing the borders of the various regions.

In the case of laminated magnetic cores, the shape of the magnetic sheets and of the inter-sheets insulation raises problems, as there is a **strong dimensional disproportion** between the length of the magnetic core and the thickness of the «leaves», as presented in the sections below.

Need for simplification …

It is therefore necessary to simplify the studied structure, as the taking into consideration of the full geometry of a structure is very expensive, almost impossible in some cases.

Simplification …

A first solution for simplification consists in building up an **equivalent macroscopic model** by means of a homogenization technique.

Homogenization is a technique which permits to simplify the behaviour law of a structure or of a material. Starting from a heterogeneous structure consisting of several materials or defects, a homogeneous equivalent structure can be determined.

What is proposed / description

The modelling of the pseudo-anisotropy associated with lamination is proposed in Flux by means of a homogenization technique.

The region of the type **Laminated magnetic non conducting region** permits the modelling of the metal sheet packets by taking into account :
- the pseudo-anisotropy (anisotropy due to lamination)
- the saturation (non linear model)

Example

Modelling of the FeSi NO (Non Oriented) magnetic sheets for motors

Compatibility

With the previous versions, the modelling of the pseudo-anisotropy was carried out by means of the customized version « Lamination 3D ». With V11.2, it is the new type of region **Laminated magnetic non conducting region fulfils this purpose.**.

4.1.3. Modelling of the pseudo-anisotropy: new type of region (2)

Introduction

This section explains the **conditions of use** for the new region **Laminated magnetic non conducting region** and gives the operating modes in 2D and in 3D.

Working hypotheses / reminder

The working hypotheses are the following :

- the hysteresis phenomenon is ignored
- the eddy currents are ignored

It is presumed, on one hand, that hysteresis does not fundamentally modify the repartition of the magnetic flux, and, on the other, that lamination fully plays its role of reducing eddy currents.

The computation of the magnetic field distribution is carried out in static (MS) , without taking into consideration the eddy current, and losses are calculated after that by means of the Bertotti formulas or the LS model, starting from the distribution of the magnetic flux density.

Modelling conditions

The modelling conditions in Flux are presented in the table below.

Domain	2D plane, 3D, Skew (?)
Physical applications	Magnetic applications: Magneto Static (MS) / Transient Magnetic (MT)
Geometric description	The magnetic sheet package is described as a homogeneous block
Physical description	• Physical region: the physical region is of the type **Laminated magnetic non conducting region** • Material*: linear or non linear approximation isotropic model

* The material models compatible with the use of this region are the following models (isotropics):

- Linear isotropic
- Isotropic analytic saturation
- Isotropic analytic saturation + knee adjustment (arctg, 3 coef.)
- Isotropic spline saturation

Limitations

The current limitations are as follows :

- The method cannot be used in the Steady State AC Magnetic (MH) applications (because of the use of the equivalent characteristics B(H))
- It is not possible to take into consideration the crystalline anisotropy*, at the same time with the pseudo-anisotropy

* The anisotropic models are not compatible with the non conducting laminated magnetic region.

Continued on next page

Result postprocessing

The postprocessing results are as follows:

- Repartition / distribution of magnetic field (B and H)
- A posteriori computation of losses:
  - Bertotti (applications MT)
  - Modèle LS (application MT)

Operating mode (2D)

The construction of the Flux project is carried out in a « standard manner ».
The specific operations are described in the table below.

Step	Action
1	Create the surface region … • Enter the name of the region : **REGION_ NAME** • Choose the type : **Laminated magnetic non conducting region** • Choose the material : **MATERIAL_NAME** • Enter the value corresponding to the thickness of sheets • Enter the value of the fill factor

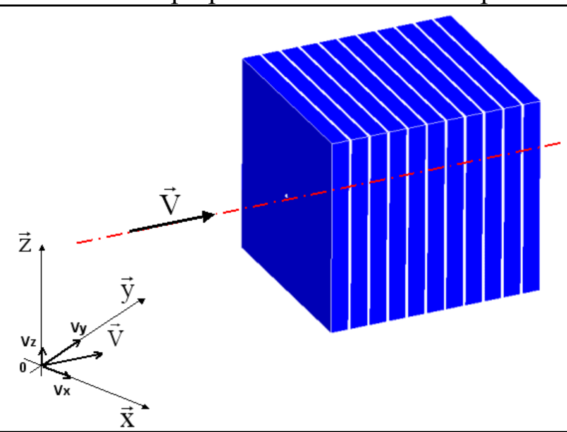
Continued on next page

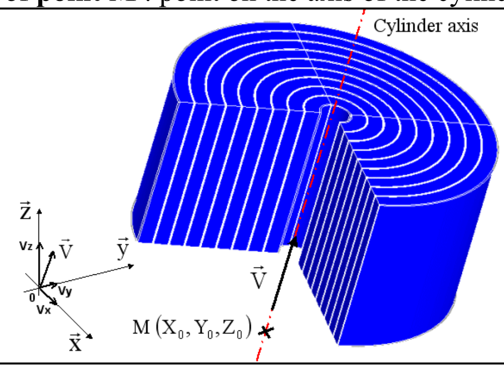
Operating mode (3D)

The construction of the Flux project is carried out in a « standard manner ». The specific operations are described in the table below.

Step	Action
1	Create the volume region …
1a	• Enter the name of the region : **REGION_NAME** • Enter a possible comment
1b	In the tab • Choose the type : **Laminated magnetic non conducting region**
1c	In the sub-tab **General** • Choose the material : **MATERIAL_NAME** • Enter the value corresponding to the thickness of sheets • Enter the values of the fill factor

1c	In the sub-tab **Geometry**
	• Choose the type of lamination :
	  • Package of flat metal sheets
	  • Package of cylindrical sheets
	• Define the direction of the sheets
	  • Acc. with following sections

Plane sheet package / direction of lamination …	
Definition within global coordinate system: of **vector V** perpendicular to the sheet plan	$\vec{V} = V_X\vec{x} + V_Y\vec{y} + V_Z\vec{z}$
	If the vector V is parallel to the Y axis Vx = 0, Vy = 1, Vz = 0

Cylindrical sheet package / direction of lamination …	
Definition within global coordinate system: of **vector V**: direction of the axis of the cylinder of **point M** : point on the axis of the cylinder	$\vec{V} = V_X\vec{x} + V_Y\vec{y} + V_Z\vec{z}$
	If the vector V is parallel to the Z axis Vx = 0, Vy = 0, Vz = 1

4.1.4. Modelling the crystalline anisotropy in 2D: new models of materials (1)

Introduction

This section serves to review the difficulties in modelling anisotropy and present the new models proposed for the B(H) non linear anisotropic characteristics.

Regarding the law of **B(H)magnetic** behaviour**,** the modelling of the anisotropy is presented in detail in the User guide (see Volume 2 / Chapter 1 « Materials principle » / §1.2.4 ; « Isotropic/ anisotropic environment (soft materials) This information is reviewed below.

Difficulties in modelling magnetic anisotropy: reminder

For an anisotropic material, permeability appears as tensor quantity with each of the components depending on the applied field.

The vector relationship B(H) is therefore written in the form of 3 families of curves:

- Bx (Hx, Hy, Hz)
- By (Hx, Hy, Hz)
- Bz (Hx, Hy, Hz)

The description of this type of curve at the experimental level is quite delicate, as one must be able to measure simultaneously both the magnitude and direction of magnetic flux density as a function of the field H. Generally, we limit ourselves to measurements in the directions where the magnetic field strength H and the magnetic flux density B are parallel.

Supposing that these characteristics exist, one must then be able to input them into the software, and the software must be able to carry out the necessary interpolations during the numerical calculations.

For all these various reasons, at the present time:

- there are **simplified models**, but whose **validity domains** are limited
- the nonlinear anisotropic models require more research

Simplified models: reminder

Generally, simplified models are based on a separation of the phenomena along the main axes of the material.

The methodology consists of expressing the vector relationship B(H) starting from **three main directions** with respect to the modelled material and achieving the most representative interpolation (linear interpolation, elliptical, …).

Each class of materials must be examined in a different manner (soft magnetic materials, permanent magnets).

Continued on next page

… in Flux: V11.1

The simplified model provided in Flux **for the linear model** incorporates the **separation of axes** with **linear interpolation**.

The vector dependence between $\vec{B}$ and $\vec{H}$ which is written as: $\vec{B} = [\mu(H)] \, . \vec{H}$ can therefore be expressed in the form of three curves: $B_x(H_x)$, $B_y(H_y)$ and $B_z(H_z)$. The correspondent axis Ox, Oy and Oz are called the main axis of magnetization.

The permeability tensor is written: $[\mu] = \begin{bmatrix} \mu_x & 0 & 0 \\ 0 & \mu_y & 0 \\ 0 & 0 & \mu_z \end{bmatrix}$

This purely mathematical formalism is very simple and convenient at the level of resolution. It renders the fact that at the microscopic level the magnetic flux density and the magnetic field strength are collinear on three main directions: privileged direction, transversal direction and a third direction.

News V11.2

The model used in Flux to take into consideration the linear anisotropic materials cannot be generalized for the non linear case.

A new model with **elliptical interpolation** has therefore been created for the non linear materials (in 2D). It is not presented in detail in this document, but the bibliographic references are given in the section below.

[Biroet al., 2010] Biro, O., Außerhoger, S., Preis, K., and Chen, Y. (2010). A modified elliptic model of anisotropy in nonlinear magnetic materials. *COMPEL : The Internationnal Journal for Computation and Mathematics in Electrical and Electronic Engineering*, 29(6):1482–1492.

[Dedulle et al., 1990] Dedulle, J.-M., Meunier, G., Foggia, A., Sabonnadiere, J.-C., and Shen, D. (1990). Magnetic fields in nonlinear anisotropic grain-oriented iron-sheet. *IEEE Transactions on Magnetics*, 36(2):524–527.

[Napoli and Paggi, 1983] Napoli, A. D. and Paggi, R. (1983). A model of anisotropic grain-oriented steel. *IEEE Transactions on Magnetics*, 19(4):1557–1561.

4.1.5. Modelling of the crystalline anisotropy in 2D: new models of materials (2)

Introduction

This section explains the **conditions of use** for the following new non linear anisotropic models :

- saturation anisotropic analytical (arctg, 2 coef.)
- saturation anisotropic analytical + control of elbow (arctg, 3 coef)
- saturation anisotropic spline

Modelling conditions

The modelling conditions in Flux are presented in the table below.

Domain	2D plane
Physical applications	Magnetic applications: Magneto Static (MS) / Transient Magnetic (MT)

Limitations

The current limitations are as follows :

- the method cannot be used in Steady State AC Magnetic applications (MH) (because of the use of the B(H) equivalent characteristics)
- it is not possible to take into account the crystalline anisotropy at the same time with the pseudo-anisotropy

Models for soft materials

The various available models for the description of soft magnetic materials are presented in the following table. The new models appear against a coloured background.

Isotropic/anisotropic material: linear approximation	
Linear isotropic $B = \mu . H$	Linear anisotropic $\vec{B} = [\mu] . \vec{H}$
Linear isotropic complex	Linear anisotropic complex

Isotropic/anisotropic material: nonlinear approximation	
Isotropic saturation: $B = \mu(H) . H$ • analytic • analytic + knee adjustment • spline	Anisotropic saturation * : $\vec{B} = [\mu(H)] . \vec{H}$ • analytic* • analytic + knew adjustment* • spline*
Parabola isotropic +straight line: $B = \mu(H) . H$	

* The regions compatible with the anisotropic models are as follows :

- non conducting magnetic regions
- regions of the solid conductor type

4.2. User sub routine (Groovy)

Introduction

This part deals with the user sub routines written in Groovy and use in Flux.

Contents

This section contains the following topics:

4.2.1. User sub routines (Groovy) : What is that ?

Introduction

User sub routines allow to extend the scope of application of Flux.
They provide the opportunity to define new physical properties (sources, materials,…) based on settings chosen by user.
Thus, user is able to model non-standard physical phenomena.

Type of customizable entities

User sub routines allow to customize three type of entities in Flux :

- I / O parameters, using the file "**PhysicalParameter.groovy**"
- non-hysteretic properties of materials, using the file "**MaterialLaw.groovy**"
- hysteretic properties of materials, using the file "**HysteresysMaterialLaw.groovy**"

Groovy

User sub routines are written in Groovy. This is the name of an object oriented programming language intended to the Java platform.
Groovy uses a Java-like syntax and is directly compiled into bytecode by a Java compiler.
As this bytecode is the same used by Java, Groovy is entirely compatible with the JVM and so it can:

- use the Java library
- be used into Java classes

For futher informations, the reader can visit the official web site:
http://groovy.codehaus.org/

Templates and Groovy examples

Once Flux is installed :

- templates of Groovy files are stocked by default in the directory:
  C:\Cedrat\Flux_11.2\Resources\UserFunction\Template
- three examples using Groovy files into Flux projects are available in the directory:
  C:\Cedrat\Flux_11.2\Resources\UserFunction\Examples

4.2.2. User sub routines (Groovy): What for?

Sub routines before the 11.2 version

Before the Flux 11.2 version, user had to write user sub routines in Fortran and compile them using a Fortran compiler.
Moreover, he also had to define an user version in Flux.

Sub routines from the 11.2 version

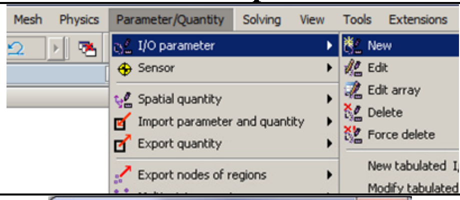
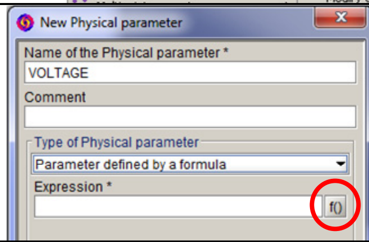
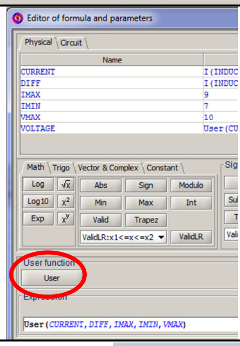
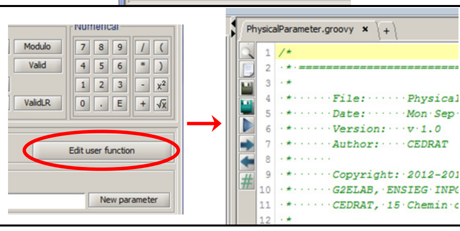
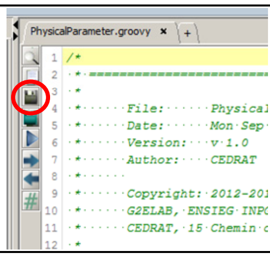
Nowadays, there is no need anymore either to buy a Fortran compiler or to create an user version.
The use of user sub routines is hence simplified.
Indeed, if user has to perform any modification in his Groovy file, all he has to do is to save it and the Groovy file will be automatically compiled during its next use.

Use of old sub routines (Fortran)

It is still possible to use old user sub routines written in Fortran. To do this, user needs to go to options/access paths/user version from the supervisor.
Then user selects the directory in which are stocked the directory which the suffix is ".f3d_usr".

4.2.3. User sub routines (Groovy): How to proceed?

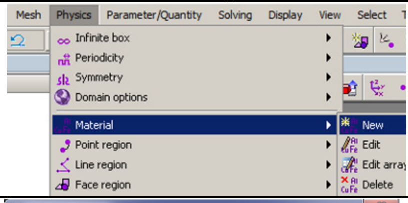
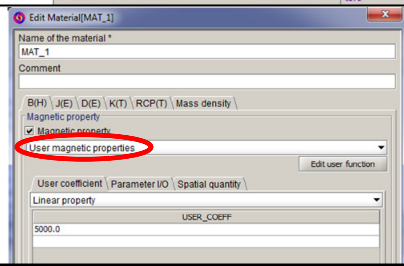
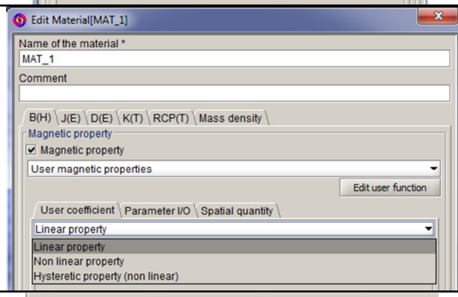
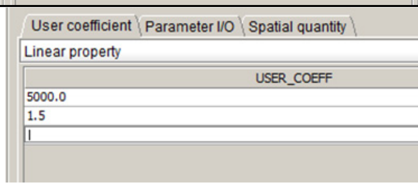
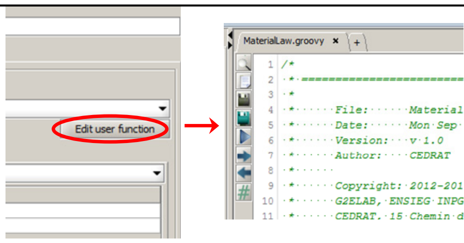
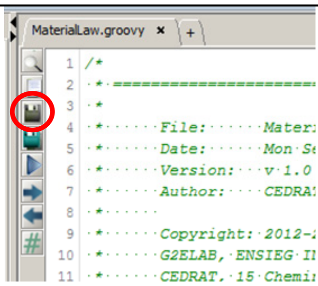
I / O parameters: operating This is the different stages to use an user sub routine to model an I / O parameter.

Stage	Description	Example
1	User creates a new I / O parameter.	
2	User selects : • a name for the physical parameter • parameter defined by a formula Then, he clicks on the editor of formula.	
3	User clicks on the "User" button. Then, he may enter the arguments that he wants into the "User" function.	
4	Clicking on the "Edit user function" button, Flux opens the file "PhysicalParameter.groovy" into the Python editor.	
5	User may code the program he wants.	
6	Once his program is written, user has to save his file. Then, the Groovy file is stocked in the current Flux project (…\persistent\groovy).	

Continued on next page

Material properties : operating

This is the different stages to use an user sub routine to model the properties of a material.

Phase	Description	Exemple
1	User creates a new material.	
2	User selects : • a name for the material • the law he wants to code • the user property of this law	
3	Then, user selects the type of his material: • linear • non linear • hysteretic	
4	Then, user selects the number and the values of : • user coefficients • I / O parameters • spatial quantities	
5	When user clicks on the "Edit user function" button, Flux opens into the Python editor, the file: • "MaterialLaw.groovy" for a non-hysteretic property • "HysteresysMaterialLaw.groovy" for an hysteretic property	
6	User may code the program he wants.	
7	Once his program is written, user has to save his file. Then, the Groovy file is stocked in the current Flux project (…\persistent\groovy).	

4.2.4. User sub routines (Groovy): limitations / advices

Limitations

Here are some limitations :

- For the user sub routines modeling the properties of materials, it is essential to use a single core in Flux. To do this, user needs to go to options/system/multi-cores from the supervisor.
- For this version, the number of arguments of the "User" function can not exceed 8.

Advices

Here are some advices:

- It is possible to save considerably computation time giving the type of the numerical data in Groovy. For instance, it is better to write 2.0d than 2.0.
- The indexation of the tables starts to 0 in Groovy.

4.2.5. User sub routines (Groovy): annexe

Physical Parameter .groovy

Here are the settings of "PhysicalParameter.groovy":

Variable	Type	Size	Informations
inputValues	Double[]	0:n-1	The n arguments of the "User" function.
outputValue	Double		

Material Law.groovy

Here are the settings of "MaterialLaw.groovy":

Variable	Type	Size	Informations
material	String		The name of the material
userCoeff	Double[]	n	The n user coefficients
paramSpacialCurrent	Map* [k,v]	n	The n spatial quantities
paramEvolCurrent	Map [k,v]	n	The n I / O parameters
nbReal	Integer		This points out if the problem is real (1) or complex (2)
nbComponant	Integer		This points out if the variable is scalar (1) or vector (3)
varin	Double[][]	nbReal x nbComponant	This is the input variable, computed by Flux
varou	Double[][]	nbReal x nbComponant	This is the output variable, computed by the user model
tensor	Double[][][]	nbReal x nbComponant x nbComponant	This is the output tensor, computed by the user model
iostatus	Integer		This points out an error to the user, if its value is different from 1 or 2

* For each value v, there is an appropriate key k. k represents the name of the spatial quantity and this map is used as follows: paramSpacialCurrent ["k"] = v.

Continued on next page

Hysteresys Material Law.groovy

Here are the settings of “HysteresysMaterialLaw.groovy”:

Variable	Type	Size	Informations
material	String		The name of the material
userCoeff	Double[]	n	The n user coefficients
paramSpacialCurrent	Map [k,v]	n	The n spatial quantities
paramEvolCurrent	Map [k,v]	n	The n I / O parameters
step	Integer		This points out the step of the computation process (1 to 8)
arrayDSize	Integer		This points out the size of the arrayD table
arrayISize	Integer		This points out the size of the arrayI table
arrayD	Double[]	arrayDSize	Work table
arrayI	Integer[]	arrayISize	Work table
coordPoint	Double[]	3	The coordinates of the Gauss point
times	Double[]	2	This points out the time step : • times[0] : current • times[1] : previous
nbReal	Integer		This points out if the problem is real (1) or complex (2)
nbComponant	Integer		This points out if the variable is scalar (1) or vector (3)
varin	Double[][]	nbReal x nbComponant	This is the input variable, computed by Flux
varou	Double[][]	nbReal x nbComponant	This is the output variable, computed by the user model
tensor	Double[][][]	nbReal x nbComponant x nbComponant	This is the output tensor, computed by the user model
iostatus	Integer		This points out an error to the user, if its value is different from 1 to 8

4.3. Generalized Bertotti model

Introduction

The Bertotti model allows computing the iron losses after the solving, for steady state and transient application.
In V11.2, the user has the possibility to adjust more coefficients of the theorical formula , the exponents.

Contents

This section contains the following topics:

Reading advice

For complementary information see the following documents :

- *User's guide volume 2, paragraphe 5.2,* ***« Computation of the magnetic losses by means of the formulas of Bertotti »***
- *Scientific paper****, « An improved approach to power losses in magnetic lamination under non sinusoïdal induction waweform »*** *- F. Fiorillo and A. Nokinov – IEEE Trans. on Magn. Vol 26 n°5 sept. 1990*

4.3.1. Bertotti model in Steady State AC Magnetic

Steady state AC magnetic : intial Bertotti

Before V11.2.0 Bertotti model was the following.

With a sinusoidal flux configuration, the total losses volumic density is :

$$dP_{TOT} = \underbrace{k_h B_m^2 f}_{\text{Hysteresis losses}} + \underbrace{\pi^2 \frac{\sigma d^2}{6}(B_m \cdot f)^2}_{\text{classic losses}} + \underbrace{k_e (B_m \cdot f)^{3/2}}_{\text{Excess losses}}$$

- k_h : Hysteresis losses coefficient
- B_m : Maximal flux density
- f : Frequency
- σ: material conductivity
- d : thickness of lamination
- k_e : Losses in excess coefficient → $k_e = \sqrt{\sigma G V_0 S}$
  - G : constant without dimension
  - S : section of lamination
  - V_0 : constant field depending from the difference of coercitive between 2 MO (magnetic objects according to Bertotti Theory)

For each region where the calculus is applied, the user must give the following coefficients :

- k_h : Hysteresis loss coefficient in $W/T^2/s/m^3$
- σ: Material conductivity en $S \cdot m^{-1}$
- k_e : Loss in excess coefficient in $W/(T.s^{-1})^{3/2}/m^3$
- d : thickness of lamination in m
- k_f : stacking factor (about 1) (useful to calculate the total iron volume)

the losses volumic density become :

$$dP_{TOT} = \left[k_h B_m^2 f + \sigma \frac{\pi^2 d^2}{6} (B_m.f)^2 + k_e (B_m.f)^{3/2} \right] k_f$$

Before the V11.2.0, the exponents are imposed and no free

Continued on next page

Steady state AC magnetic : generalised Bertotti

From the 11.2.0 version, exponents are free and can be modified by the user.

$$dP_{TOT} = \left[k_h B_m^{kb1} f^{kf1} + \sigma \frac{\pi^2 d^2}{6} B_m^{kb2} f^{kf2} + k_e B_m^{kb3} f^{kf3} \right] k_f$$

With the coefficients that user must fill in :

k_h : Hysteresis loss coefficient in $W/T^2/s/m^3$
σ : Material conductivity en $S \cdot m^{-1}$
k_e : Loss in excess coefficient in $W/(T.s^{-1})^{3/2}/m^3$
d : thickness of lamination in m
k_f : stacking factor (about 1) (useful to calculate the total volume)
k_{b1} : B exponent for hysteresis loss term
k_{f1} : f exponent for hysteresis loss term
k_{b2} : B exponent for classical loss term
k_{f2} : f exponent for classical loss term
k_{b3} : B exponent for losses in excess term
k_{f3} : f exponent for loss in excess term

By default, these 6 exponents are set up to the initial values of Bertotti Model :
$k_{b1} = 2$ $k_{f1} = 1$ $k_{b2} = 2$ $k_{f2} = 2$ $k_{b3} = 3/2$ $k_{f3} = 3/2$

New tool : determination of coefficients

An Excel sheet permitting to determine coefficients and exponents of Bertotti formulation is available. This tool has been stored on our space disk, during the installation, at the following path :
C:\Cedrat\FluxDocExamples_11.2\Tools\BertottiLossesCoefficients

(If Flux is installed a the default path C:\Cedrat)

4.3.2. Bertotti model in Transient Magnetic

Magnetic Transient : initial Bertotti

Before the V11.2.0, the Bertotti model was the following. The instantaneous losses volumic density is writing as following :

$$dP_{TOT}(t) = \underbrace{k_h\, P_h(t)}_{\text{hysteresis losses}} + \underbrace{\frac{\sigma d^2}{12}\left[\dot{B}(t)\right]^2}_{\text{classical losses}} + \underbrace{k_e\left[\dot{B}(t)\right]^{3/2}}_{\text{Losses by Excess}}$$

- k_h : Hysteresis loss coefficient
- k_e : Loss in excess coefficient in $W/(T.s^{-1})^{3/2}/m^3$

For each region where the calculus is applied, the user must give the following coefficients :

- k_h : Hysteresis loss coefficient in $W/T^2/s/m^3$
- σ : Material conductivity en $S\ .\ m^{-1}$
- k_e : Loss in excess coefficient in $W/(T.s^{-1})^{3/2}/m^3$
- d : thickness of lamination in m
- k_f : stacking factor (about 1) (useful to calculate the total volume

the volumic density losses become :

$$dP_{TOT}(t) = \left[k_h dP_h(t) + \sigma\frac{d^2}{12}\left(\dot{B}(t)\right)^2 + k_e\left(\dot{B}(t)\right)^{3/2}\right]k_f$$

avec $dP_h(t) = B_{max}^2 \cdot f$

In the case of step by step, the main interest is to give the mean value of the instantaneous losses density, during a period for example. In this case, the expression become :

Dans le cas du pas à pas, le grand intérêt est l'étude de la valeur moyenne de la densité des pertes instantanées, au cours d'une période par exemple. Dans ce cas, l'expression devient :

$$\frac{1}{T}\int_0^T dP_{TOT}(t)\,dt = k_h\,B_{max}^2\,f k_f + \frac{1}{T}\int_0^T\left[k_h\frac{d^2}{12}\left(\dot{B}(t)\right)^2 + k_e\left(\dot{B}(t)\right)^{3/2}\right]k_f\ dt$$

Where B_{max} is the maximal flux density reach during a period.

Before the V11.2.0, the exponents are imposed and no free.

Continued on next page

Transient Magnetic : generalised Bertotti

From the 11.2.0 version, exponents are freed and can be modified by the user.

$$dP_{TOT}(t)=\left[k_h dP_h(t)+\sigma\frac{d^2}{12}\left(\dot{B}(t)\right)^{kb2}+k_e\left(\dot{B}(t)\right)^{kb3}\right]k_f$$

with $dP_h(t)=B_{max}^{kb1}\cdot f^{kf1}$

with the coefficient that user must fill in :

k_h : Hysteresis loss coefficient in $W/T^2/s/m^3$
σ : Material conductivity en $S \cdot m^{-1}$
k_e : Loss in excess coefficient in $W/(T.s^{-1})^{3/2}/m^3$
d : thickness of lamination in m
k_f : stacking factor (about 1) (useful to calculate the total iron volume)
k_{b1} : B exponent for hysteresis loss term
k_{f1} : f exponent for hysteresis loss term
k_{b2} : B exponent for classical loss term
k_{b3} : B exponent for losses in excess term

the 4 supplementary exponents are the default value of the initial model of Bertotti :

k_{b1} = 2 k_{f1} = 1 k_{b2} = 2 k_{b3} = 3/2

New tool : determination of coefficients

An Excel sheet permitting to determine coefficients and exponents of Bertotti formulation is available. This tool has been stored on our space disk, during the installation, at the following path :
C:\Cedrat\FluxDocExamples_11.2\Tools\BertottiLossesCoefficients

(If Flux is installed a the default path C:\Cedrat)

5. V11.2 new features regarding Solver

Introduction

This chapter deals with the V11.2 new features regarding the solver.

Contents

This chapter contains the following topics:

5.1. Adaptive solver

Introduction

This section deals with the adaptive solver and how to use it with Flux software.

Contents

This section contains the following topics :

Other terminologies

Refinement, adaptive mesh, auto-adaptive mesh, auto-adaption of the mesh, adaptive solver, adaptive solving

5.1.1. Adaptive solver: about

Introduction

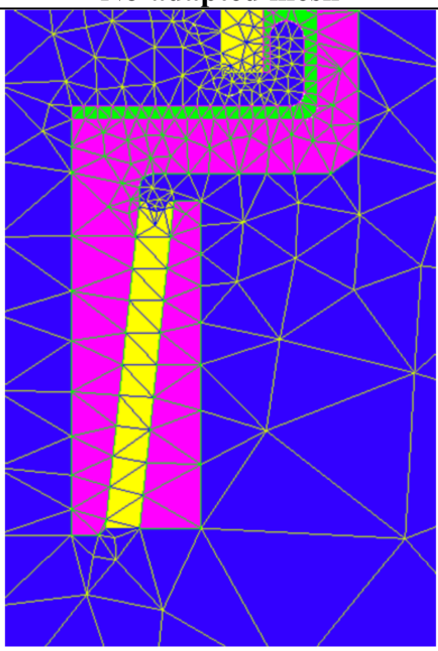
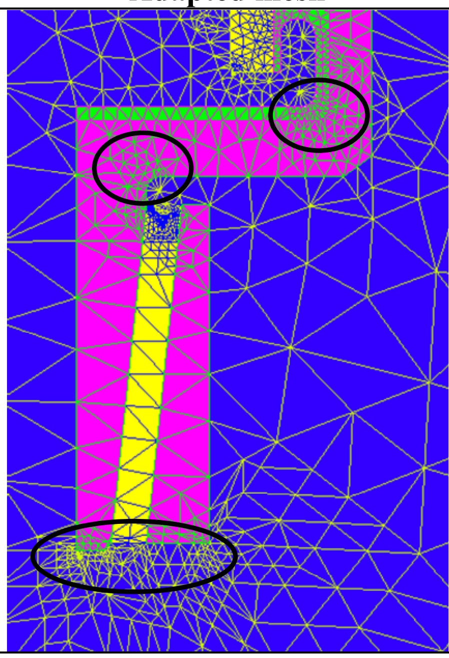
The finite element method of analysis offers a numerical solution for physical phenomena described by partial differential equations. For this, it relies on a sampling of the studied domain that is called mesh. The denser the mesh of the studied domain, the more accurate the numerical solution will be, but this is at a high cost in terms of computer memory requirements.
It is therefore essential to generate a mesh that should be refined in the zones that require it and more coarse in the rest of the domain. In other words, the mesh must be adapted, both, to the geometry and to the physics demands of the problem.

History

Over the last years, considerable effort has been made in order that the Flux software could automatically generate a qualitative mesh.
The first stage was to set up the **aided mesh,** which permits the user to obtain a mesh adapted to the geometry of the problem.
Today, the Flux software offers the **adaptive solver**. This process facilitates automatically refined mesh in the locations where the physics of the problem require it.

Description

The adaptive solver is a process that permits modeling a problem by means of an adaptive mesh. This mesh presents sizes of elements, which correspond to the local behavior of the considered physics.
In the zones requiring a finer analysis of the studied phenomenon, this translates by:
- A tightening of the meshes
- A diminution of the finite elements size

No-adapted mesh	Adapted mesh
	

Continued on next page

Limitations / Restrictions

Here are some restrictions:

- The adaptive solver is reserved to the applications:
  - Magneto Static 2D
  - Electro Static 2D
- The adaptive solver does not work with the :
  - mechanical sets of the « compressible » type
  - non conducting magnetic regions, laminations
  - anisotropic, non linear regions
- It is not advisable to use the mapped mesh with the adaptive solver
- As for all solvers, one should start from an initial mesh

5.1.2. Adaptive solver: error and shutoff criterions

Introduction

Within the framework of an adaptive solver, it is necessary to use:

- An error criterion to detect the meshing areas which are too coarse
- A stop signal criterion to set a term to the iterative process in the adaptive solver

Error criterion

In electrostatic, the error criterion is based on the Maxwell-Gauss equation: $div\vec{D} = \rho$.

In magnetostatics, the error criterion is based on the Maxwell-Ampère equation: $r\hat{o}t\vec{H} = \vec{J}$.

From a numerical viewpoint, it should be verified that the accuracy of the equations « weakly » resolved should be satisfactory.

Shutoff criterion

Currently, there are two stop signal criteria :

- The error threshold based on the energy. This permits the study of the evolution of the energy over a region by calculating the relative energy error from an iteration to another
- The maximum number of iterations of the adaptive solver

How to choose the threshold?

The value of the error threshold is defined either by the user, or automatically by Flux :

- A weak threshold (s = 0.25) is equivalent to a high accuracy
- An average threshold (s = 0.50) is equivalent to an average accuracy
- A strong threshold (s = 0.75) is equivalent to a weak accuracy

Advanced mode

In certain regions, the variation of the electromagnetic field is quasi-inexistent, i.e. null. This is the case of the infinite box, of the perfectly conducting regions and of the inactive regions. In these cases, the Flux software automatically excludes those regions from the adaptive solver.

It is also possible for the user to exclude the regions that he wishes. For this the user would go to the options box corresponding to the adaptive solver.

☞ **Solving → Adaptive solver options → Edit**

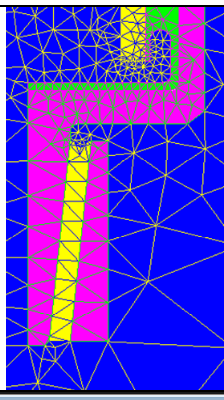
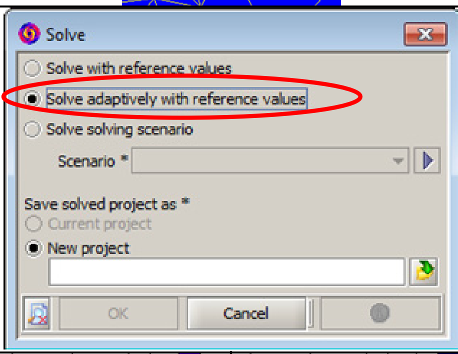
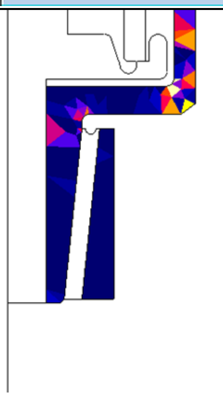
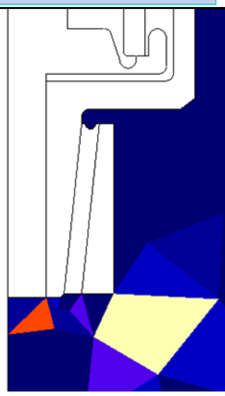
5.1.3. Adaptive solver: operating mode

Introduction

There are two methods of activating the adaptive solver:
- the user may solve a Flux project in its current state, with reference values
- Or the user wishes to solve a Flux project by means of a scenario

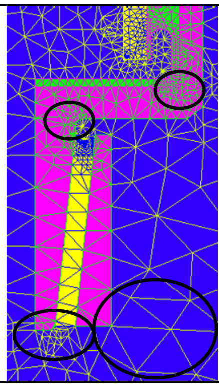
Operating mode (1)

Here are the different phases of the adaptive solver when the user solves a Flux project **in its current state**.

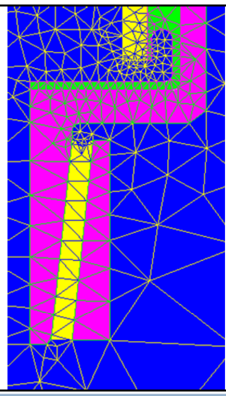
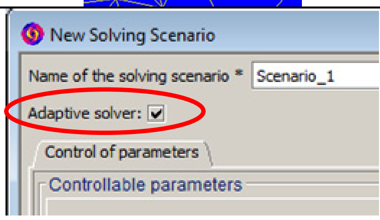
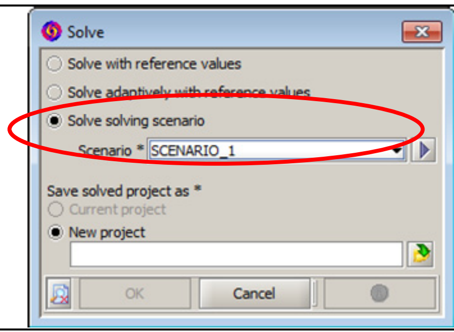
Stage	Description	Illustration
1	The user carries out an initial mesh. It is **strongly** recommended that the mesh should be carried out by means of the aided mesh. By definition, this mesh is « non-adapted » to the physics of the problem.	
2	To activate the adaptive solver, the user must first open the dialog box « **Solve** » which can be found in the « **Solving** » menu. Then, select « **Solve adaptively with reference values** ».	
3	For each assigned region, the quality of the mesh is automatically evaluated by means of an error criterion, based on one of the Maxwell equations, appropriated to the studied problem. *(a) Epoxy region* *(b) Air region*	 *(a)*  *(b)*

Continued on next page

Operating mode (1) (suite)

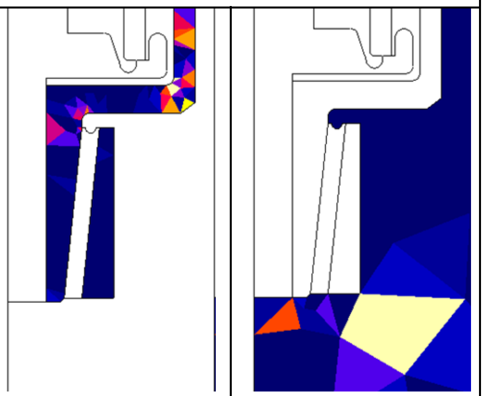
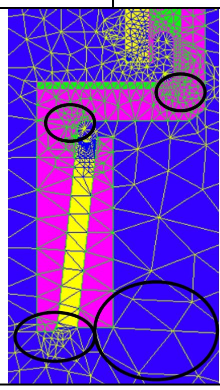
Stage	Description	Illustration
4	The mesh is then adapted to the environments where the physics of the problem requires it.	
5	The process will be repeated n times until the stop signal criteria are observed.	

Operating mode (2) Here are the different phases of the adaptive solver when the user solves a Flux project **by means of a scenario**.

Stage	Description	Illustration
1	The user carries out an initial mesh. It is **strongly** recommended that the mesh should be carried out by means of the aided mesh. By definition, this mesh is « non-adapted » to the physics of the problem.	
2	To activate the adaptive solver, the user must select « **Adaptive solver** » in the scenario that he has created.	
3	The user solves this scenario by opening the dialog box « **Solve** », which exists in the « **Solving** » menu and by selecting« **Solve solving scenario** ».	

Continued on next page

Operating mode (2) (suite)

Stage	Description	Illustration
4	For each assigned region, the quality of the mesh is automatically evaluated by means of an error criterion, based on one of the Maxwell equations, appropriated to the studied problem. *(a) Epoxy region* *(b) Air region*	 *(a)* *(b)*
5	The mesh is adapted then to the environments where the physics of the problem requires it.	
6	The process will be repeated n times until the stop signal criteria are observed.	

Use advice

For a simple, efficient use of the adaptive solver, the user can follow the next Steps:

Step	Action
1	Starting from an aided coarse mesh such as: • Relative deviation = 0.1 • Coefficient of relaxation of the line type = 0.9 • Coefficient of relaxation of the face type= 0.9
2	In a great majority of cases, two iterations are enough to obtain a mesh adapted to the physics of the problem.

5.1.4. Adaptive solver: re-mesh strategy

Introduction

In the process of adaptive solving, it is essential to determine a threshold that will include the meshing areas to be refined. This threshold depends primarily on the number of finite elements of the considered region. Two cases are then considered:

- A number of elements less than, or equal with 100
- A number of elements greater than 100

Sparse number of finite elements

In the first case, the number of finite elements is considered too sparse to establish a statistical law. The selection threshold is then defined in a controlled manner. Correspondingly, this threshold diminishes with the number of elements.

High number of finite elements

In the second case, the repartition of the number of finite elements is observed as a function of the size order of the error criterion. This permits the establishment of a logarithmic distribution law. The Gaussian law, thus obtained, is compared with the Gaussian law centered over the same interval. This allows the adjustment of the selection threshold. Thus, it depends both on the repartition and on the number of elements.

5.2. Improvement of Transient initialisation

Introduction

This section deals with new features relative to the **transient initialisation** which can used in **Transient Magnetic** and **Transient Thermal** applications. The transient initialisation allows fixing the initial conditions.
The transient initialisation can be applied by:

- static computation (already existing in Flux)
- **finite elements solution** (V11.1 new feature + improvement and finalization in V11.2)

The following documentation concerns only the initialisation by finite elements solution.

Contents

This section covers the following topics :

Reading advice

For more information on the transient initialization, please read on the user guide (available from the software) the chapter **Thermal application: Principle** and the chapter **Magnetic application: Principle**.

5.2.1. Transient Initialization: about

TM application: reminder

A **transient application** is a physical application characterized by:
- properties that vary in function of time: variable sources, variable material characteristics …
- a **time equation** (differential equation of the 1st order) : d/dt ≠ 0 (transient regime or variable regime).

The resolution consists of a sequence of resolutions in time. The solutions are not independent from each other: each solution is linked to the previous one by the temporal equation.

Initialization « at zero »

In the most general case, state variables are initialized at zero (for t = 0). The 1st step of time ($t = t_1$) corresponds to the first Flux solving step as presented in the table below.

	Initial instant	**1st time step**	**…**
Flux solver	t = 0	$t = t_1$	
B, H, …	B, H, … null everywhere*	B, H, … calculated	
Temperature	$T_0 = T_{init}$	$T_1 = T_{computed}$	

*in 3D, if there are non-meshed sources (non meshed coils or imposed field), the values of B and H are those related to these sources (Hj field).

Initialization by static computation

In the presence of magnets or sources of non null current/voltage at t = 0 s, the zero initialization in Flux is not carried out « in real accordance with the physical reality ». The starting is accompanied then by a “numeric transient”, which distorts the solving process.

To prevent from this problem, it is possible in Flux to take into account initials conditions of the **static computation initialization** type.

Initialization by FE solution

It is possible to initiate a transient computation by using as initial state the result of a :
- Magneto Static, Steady State AC Magnetic, or Transient Magnetic problem
- Steady State Thermal problem or of a Transient Thermal problem

This permits for example :
- to study a variation around an average value without being obliged to carry out all the time steps permitting to attain the steady state regime
- to resume a computation after the modification of the value of a physical size (current, convection coefficient ...)

5.2.2. Initialization by FE solution in Transient Magnetic: operation

Starting by FE solution

It is possible to start a transient computation by using as initial state the result of a Magneto Static, a Steady State AC Magnetic or a Transient Magnetic problem.

MH-TM relevance/ example

The MH applications permit the study of the steady state regime, but it has the following drawbacks :

- approximations in the taking into consideration of the magnetic saturation*
- no possibility to compute the harmonics in space and time

The TM applications compensate for the limits quoted above, but it has the drawback of requiring a computation time that can be quite long. Then, the number of time steps is important.

Starting a TM application on the basis of the MH solution permits to rapidly reach the steady state regime without being forced to solve all the time steps of the TM problem until the steady state regime is reached.

Reminder (see …):
With a Steady State AC Magnetic application, the approximations are carried out in order to take into consideration the non linear materials. We deal then with the equivalent B(H) characteristics (B sinus, H sinus, …). The local values obtained with such equivalent curves get more or less accurately close to the real result (Example of the asynchronous machine).

Relevance TM-TM / example

As with a MH application, the TM application permits to obtain the steady state regime more rapidly without the need for the user to make again the computation of the transient phases or those for startup (starting up of a synchronous machine with magnets, for instance).
It also permits to reach a certain specified point of the steady state regime – for example the passage at zero of the current of one of the phases of an electric alternator.

Finally, and mainly, it gives the possibility of an initialization by FE problem when the system comprises permanent magnets, which is not possible with an initialization by an MH problem.

Operation

The operation principle is described in the table below in the case of a MH – TM sequence.

Phase	Description
1	Construction of the Flux 1 project (MH application)
2	Solving of the Flux 1 project (MH application)
2	Export of the FE solution : Creation of an exchange file (format FTS)

Phase	Description
1	Construction of the Flux 2 project (TM application) on the basis of the Flux 1 project (MH application)
2	Modifications of the physical properties
3	Choice of the initial conditions : • Initialization of the type «by file » • Exchange file
4	Solving of the Flux 2 project (TM application)

5.2.3. Initialization by FE solution in Transient Magnetic: particularities

Starting a TM application on the basis of the solution of a MH application

In the case of a TM application starting on the basis of the solution of a MH application, it is necessary to carry out the choice of the phase at the moment of the export of the FE solution.

In the most general case (standard operation *), choose the phase $\varphi = 90°$ at the moment of the export of the FE solution.

*Reminder :

- Magneto - harmonic application:
  the sources are maximal while the phase is null (function in cosine)
- Transient Magnetic application:
  the sources are maximal while the phase has the value $\pi/2$ (function in sine).

Starting a TM application on the basis of the solution of a TM application

The principle of a TM application starting on the basis of TM solution has progressed between the previous version V11.1 and the current version V11.2:

With the previous version V11.1	Starting from the current version V11.2
In V11.1, in the case of a TM application starting on the basis of the solution on a TM application, it was necessary to take certain precautions with the cinematic coupling and/or with the circuit coupling. ***Cinematic coupling:*** At the moment of export of the FE solution : • Note the value of the speed and of the position of the mechanical set in motion (translation or rotation) • Introduce these values in the project Flux 2 (initial values of the mechanical set in motion) ***Circuit coupling:*** If the current/voltage source is described by means of a time formula, in the Flux 2 project, move back in time the current/voltage source.	Starting from the V11.2, in the case of a starting on TM solution, the second project will start automatically at step time chosen in the first project during the creation of initialization file *.FTS. In this case it is not necessary to reinitialize the second project (Circuit and mechanical set).

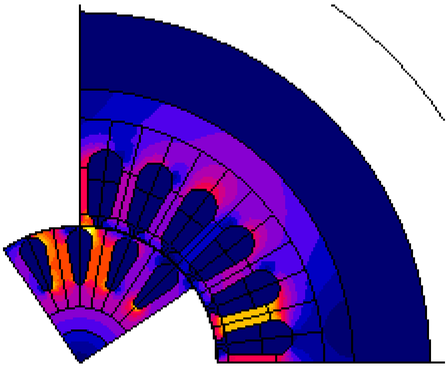
5.2.4. Initialization by FE solution in Transient Magnetic: examples

Example 1 : presentation

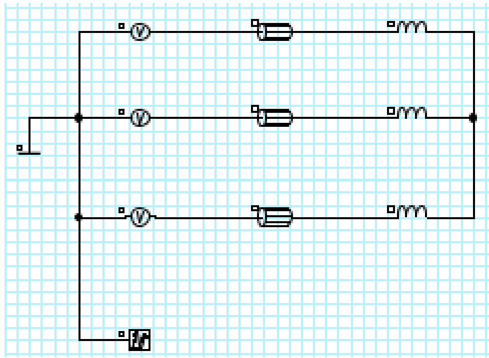
The studied device is an asynchronous motor.

Computation conditions:
A first MH computation for the slip value of 6 % is carried out. It permits to establish the corresponding steady state regime of the device. It also permits to the have the steady state field of the current density in the rotor bars.

Domain finite elements



Electrical circuit

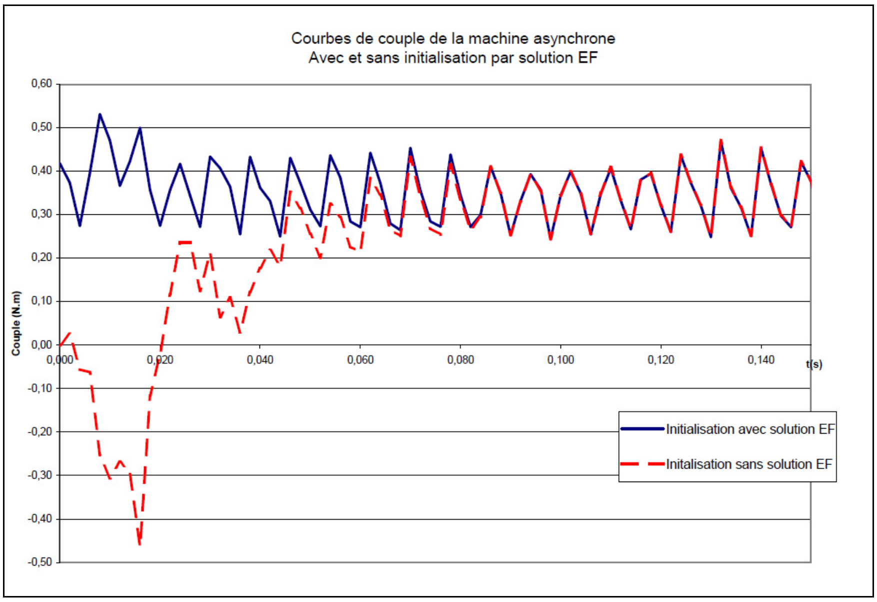


Example 1 : results

The curve of the time variation of the electromagnetic torque of the asynchronous machine is represented in the figure below in the two following conditions : with or without initialization by a FE solution.

Main results:

- without initialization by a FE solution (dotted lines) :
  one can note a transient period before the curve reaches the steady state regime
- with initialization by a FE solution (solid lines) :
  the curve reaches the steady state regime faster



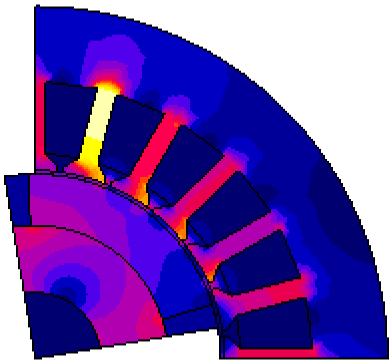
Continued on next page

Example 2 : presentation

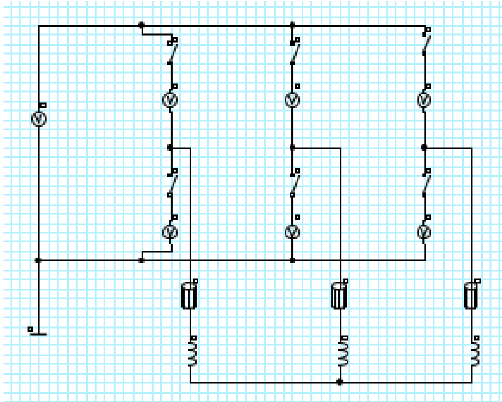
The studied device is a synchronous motor with permanent magnet.

Computation conditions :
A first TM computation is made, simulating the increase of the machine speed to the rated point (n_{nom} = 1000 rpm). The second computation, initialized by the FE solutions of the first one goes back to the steady state regime.

Domain finite elements



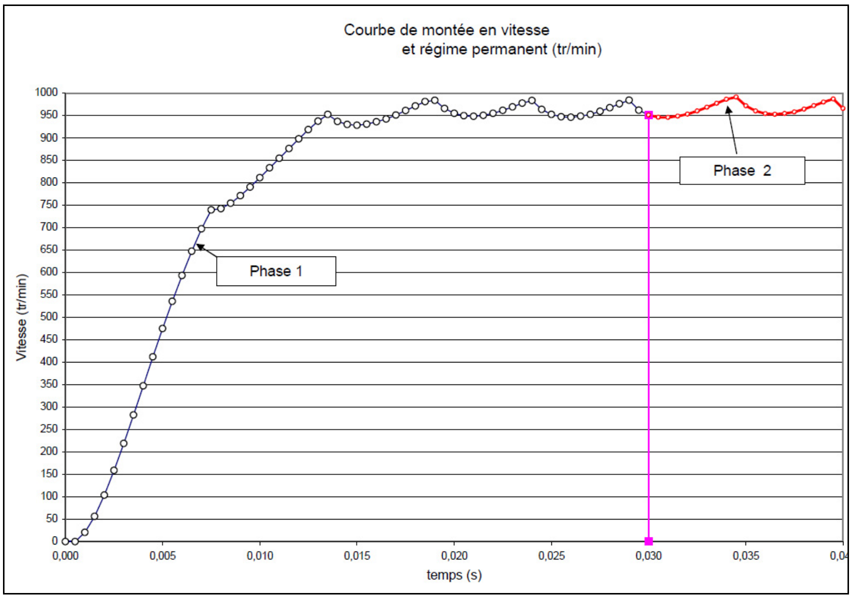
Electrical circuit



Example 2: results

The curve synchronous machine speed increase is represented below.

A first phase consists in reaching the rated speed 1000 rpm. The second problem considers the results of the first phase at the time t = 0.03 s.



5.2.5. Initialization by FE solution in Transient Thermal: operation

Starting by a FE solution

It is possible to start a new transient thermal computation by using as the initial state the result of a Steady State Thermal problem or of a Transient Thermal problem.

TT-TT relevance / example

For the applications related the thermal treatment, the study of the «cycles of heating» (heating / holding / cooling) will permit an optimization of the process.

To simulate the heating cycles two strategies are possible in Flux.

With the first strategy, (« via formula »), the inductor, as well as the physical properties of the materials are piloted by means of formulas, for the different phases of the process. It is therefore necessary to know beforehand the times of each of the phases, which is not always the case.

With the second strategy, the study of the different phases is cut out within the time (independent Flux projects) and the transfer of data between the Flux projects is carried out by file. It is this operation that is explained in the next section.

Operation

The operation principle is described in the table below for the case of a sequence : TT - TT

Phase	Description
1	Construction of the Flux 1 project (TT application)
2	Solving of the Flux 1 project (TT application)
2	Export of the FE solution: creation of an exchange file (format FTS)

Phase	Description
1	Construction of the Flux 2 project (TT application) on the basis of the Flux 1 project (TT application)
2	Modifications of the physical properties
3	Choice of the initial conditions : • Initialization of type « by file » • Exchange file
4	Solving of the Flux 2 project (TT application)

5.2.6. Initialization by FE : examples

Example 2D

2D examples are presented in the following technical document :

- **Induction heating tutorial**: study of a heating by induction of a steel sheet
- **Induction motor tutorial** : study of an induction motor (asynchronous)

Example 3D

A 3D example is presented in the following technical document:

- **Magneto-Thermal 3D tutorial**: study of the thermal treatment of a steel flange.

Exemple Skew

A SKEW example is presented in the following technical document:

- **Induction motor with Skew tutorial**: study of an induction motor (asynchronous) with rotor skewed.

5.2.7. Initialization by FE : Improvements in V11.2

Introduction

This map presents improvements about transient initialization (by solution FE) in V11.2 relative to previous version V11.1

Difference V11.1 vs V11.2

The table below shows the major difference of operation mode between the 11.1 version and the 11.2 version.

With the 11.1 version	With the 11.2 version
User had to be very cautious with the connexion between two magnetic transient problems (MT1 → MT2). Because scenario started always at $t_{initial} = 0$, when user created a transient file in a first magnetic transient problem (MT1) at t_1, physic of problem MT2 need to be adapted and shifted from this time value t_1. • Analytical equation in electrical circuit need to be shift of $\Delta t = t_1$, • Mechanical set need to be shift of $\Delta\theta = \theta_1$ and velocity had to be equal at the velocity reach in the MT1	Scenario can start at $t = t_1$, not necessarily at t= 0 s. As a result, user didn't need to change analytical equation in electrical circuit and shift the rotor position and velocity in the mechanical set. All is performed directly by Flux during the loadinf of .FTS file.

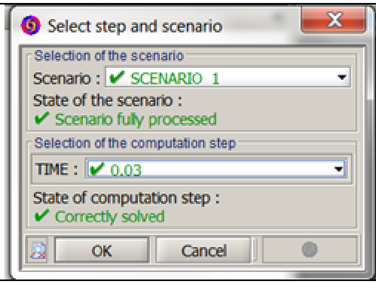
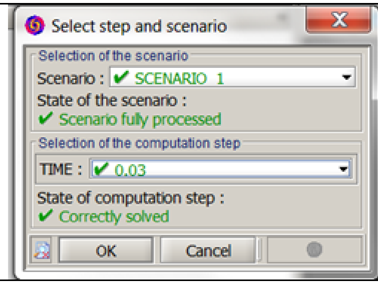
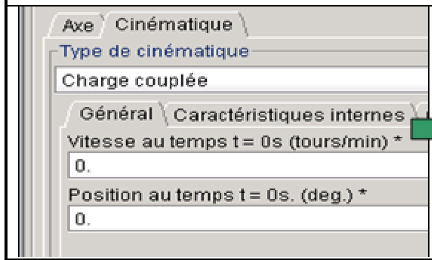
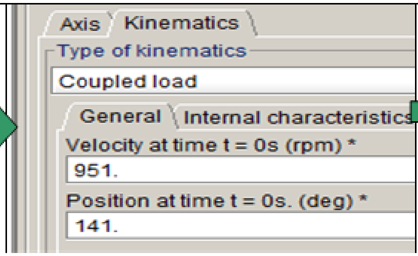
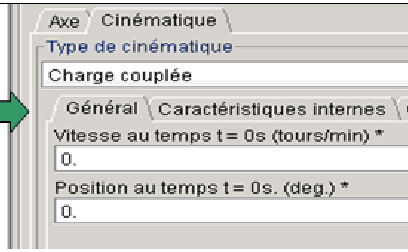
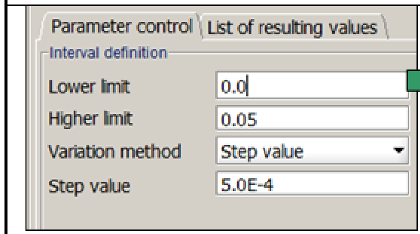
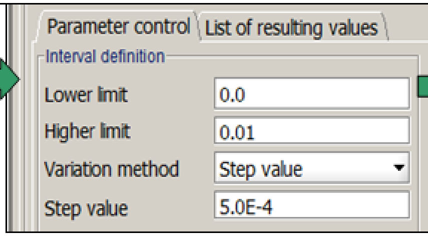
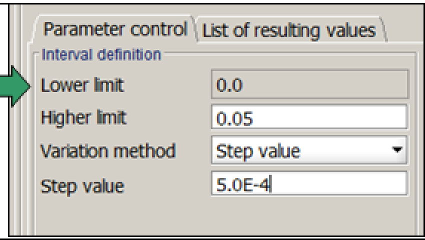
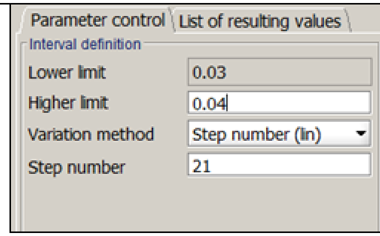
Be careful:

- Try to keep the same interval time step between the two problem or at least, not too different
- Mesh must be identical between the two projects.

Example

The table below presents an example which shows the improvements added in 11.2 version relative to 11.1 version.

Continued on next page

11.1.0		11.2.0.	
MT 1	MT 2	MT 1	MT 2
As an example : Transient fil .FTS is created at t = Δt_1 = 0.03 s		As an example : Transient fil .FTS is created at t = Δt_1 = 0.03 s	
Electrical circuit		Electrical circuit	
$\boxed{V(t) = V_{MAX} * \sin(w*t+\varphi)}$	$\boxed{V(t) = V_{MAX} * \sin(w*(t+\Delta t_1)+\varphi)}$	$\boxed{V(t) = V_{MAX} * \sin(w*t+\varphi)}$	$\boxed{V(t) = V_{MAX} * \sin(w*t+\varphi)}$
Mechanical set		Mechanical set	
			No change
Scenario solving		Scenario solving	
			

5.3. Improvement of MUMPS solver

Introduction

In the framework parallel calculus technology deployment Mumps was integrated in FLUX with two distinct versions :

- Mumps : direct solver
- Mumps distributed (beta version)

Contents

This section contains the following topics :

5.3.1. Solving with Mumps

Introduction

MUMPS (« **MU**ltifrontal **M**assively **P**arallel **S**olver ») give the possibility to solve linear equation system like Ax = b with A, a parse Matrix. MUMPS has a direct method base on Multifrontal approach.

With Linux 64, MUMPS direct solver is not available for steady state application. Use MUMPS distributed version.

Multi processus Multi –Threads Calculations

Mumps works with two complementary parallel technologies :
- MPI (Message Passing Interface) using for Multi-Processus calculations
- MKL (Math Kernel Library) using for Multi-Thread calculations.

Mumps direct *sequential*

Solver « Mumps : direct solver » is mono process but multithread. There is only one **MPI** processes while number of thread is fixed by cores number choose by the user from the supervisor.

Mumps *Distributed*

« Mumps distributed : direct solver »is :
- Multi-process
- Multithread

Processus and thread number are automatically specified by Flux. Algorithm takes in count the core number specified in the supervisor.

Comparative sequential / distributed

To resume :

	Mumps direct solver (sequential)	**Mumps distributed**
Processus	Only one	Several (*Number automatically determined by Flux*).
Thread	Several (*Number equal to the cores number specified by the user*)	Several (*Number automatically determined by Flux*).

5.3.2. Memory consideration with MUMPS

Principle

By default, Mumps use all the memory (RAM) available.
If this is not the case, Mumps will use :
- Either the swap file which slow down the calculation,
- Or the disk memory.

Practical aspects

.
By default, in the case of project with a matrix size

Less than 300 000 row :

In the case where the RAM becomes insufficient, Mumps will use the swap file and calculation will be slow down.
If this first solution is insufficient, Mumps will use automatically the Out Of Cores (OOC) mode
More than 300 000 row : OOC mode is automatically choose …

Be careful

RAM using by MUMPS is independent from the memory specified in the supervisor.

Advice

Using a hard disk with SSD technology is recommended to obtain good performances

Option modification, users choices

When a project use in the same time the swapt file and RAM memory, calculations slow down. Choose the OOC mode will be better.

Transition In Out Of Cores (OOC) mode

In the menu **Solving** / **Solving process options** / **Edit**, in the MUMPS tab, user can impose the OOC mode.
The temporary files (delete at the end of the solving) are written by default in the tmp directory. Alternatively, U
user can specify a directory of his choice.

5.4. Improvement of Flux Skew

Introduction

This section deals with the improvement of Flux skew application
Improvement works of Flux Skew have been implemented on version 11.1.
This Version 11.2 has permitted to finalize the work involved.

Contents

This section contains the following topics:

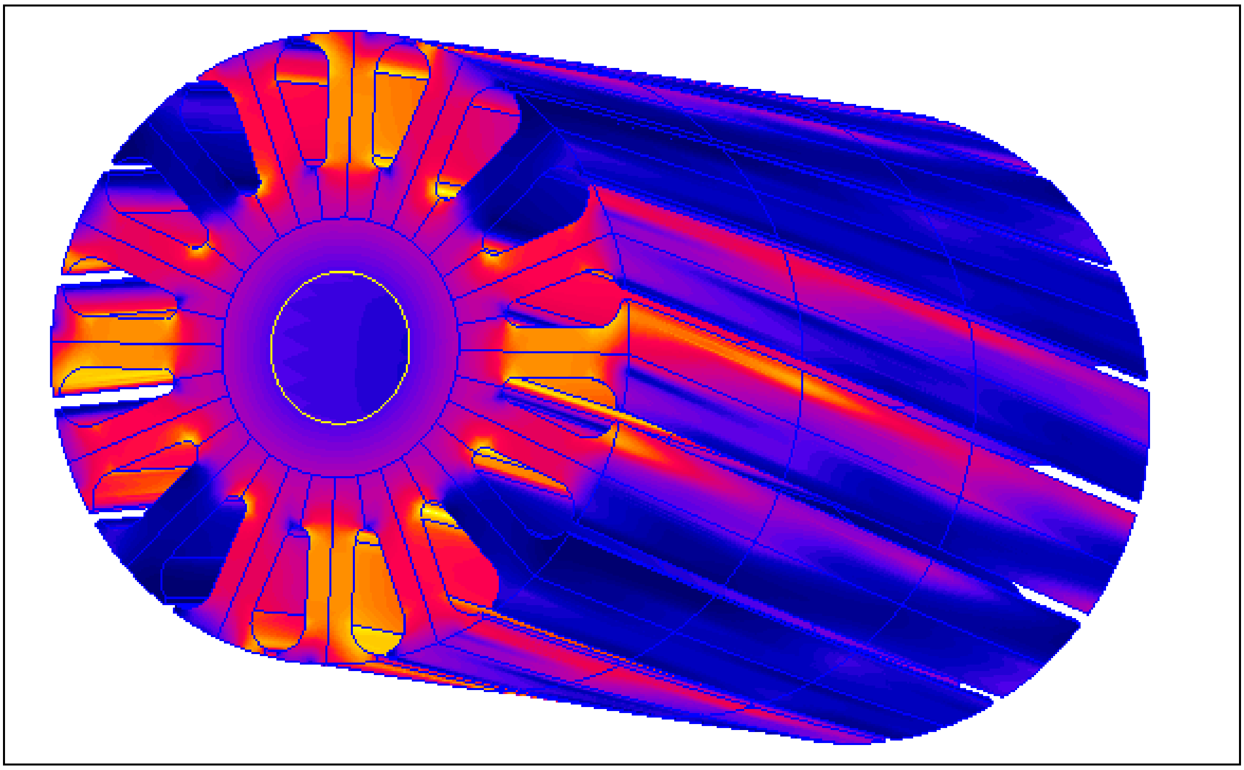
5.4.1. Improvement of Flux Skew in V11.1

Introduction

This paragraph is a reminder of improvement works of Flux Skew achieved in the previous version.

Postprocessing optimization

Since the V11.1, the postprocessing of a Skew project can be displayed in 3D. The device with layers is rebuilt after the solving process finished.
It permits to display all graphical results on the full device (on modelled layers + between layers).



Use advice:
If the user wishes benefit of the 3D optimal postprocessing for a project already solved with an earlier version to V11, the user must delete the results and restart the solving process.

Some improvements

Some improvements have been added in V11.1 :

- the sliding is not defined any more in the application but directly in the mechanical set

the parametric study is possible with “physical” parameters (not available with « geometric » parameters)

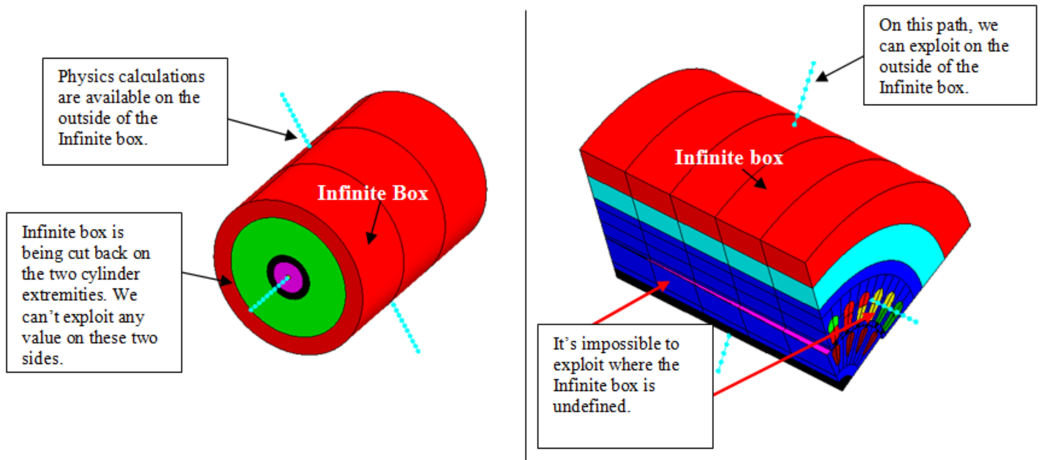
5.4.2. Improvement of Flux Skew in V11.2

Introduction

In the continuity of improvement works about Flux Skew achieved in V11.1, the version 11.2 of Flux some points about this application:
- Infinite box capability has been added
- Geometic parameter is now available
- Display of volume regions after solving
- 3D geometry building speed increased prior to the solving process
- «Stop / Resume solving » management

Infinite box

It is now possible to add an infinite box to a project which permits the user to calculate physical quantities in the air region outside the studied device on different spatial supports
In a SKEW application, the Infinite box is of the truncated cylindrical type (see the illustration below).



Project without periodicity Project with periodicity.

Attention : The on a point outside the Infinite box is impossible.

Geometric parameter

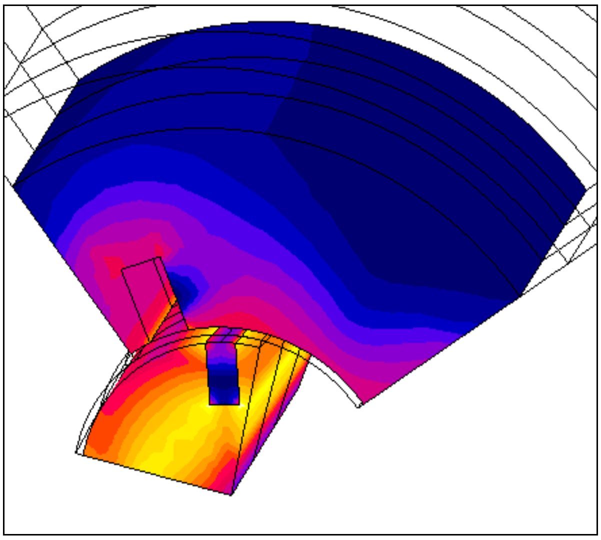
Studies with geometric parameters have been implemented in the Flux version V11.2.0 for the SKEW applications. The user can make a geometric parametric analysis of a problem. The main advantage of a parametric analysis is the ability to take into account several configurations of the studied device in one Flux project. The influence of a specific parameter can be directly visualized in the postprocessing results.

We use the terms :
- **Multiparametric analysis,** with a Magnetostatic (MS) or a Steady State AC Magnetic (MH) application, when the user is interested in the influence of the geometric and/or physical parameters.
- **Parametric transient analysis,** with a Transient Magnetic (MT) application, when the user is interested in the influence of the geometric and/or physical parameters, beside the TIME parameter.

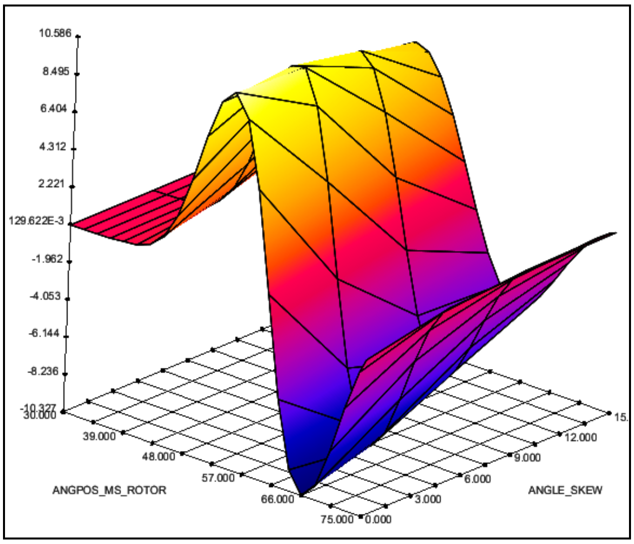
Continued on next page

Example of parametric study

Here is an example, with a simplified geometry, permitting the influence of the twisting angle on the electromagnetic torque to be evaluated:



Geometry (electrical machine with only one slot on the quarter of the rotor and stator armatures)



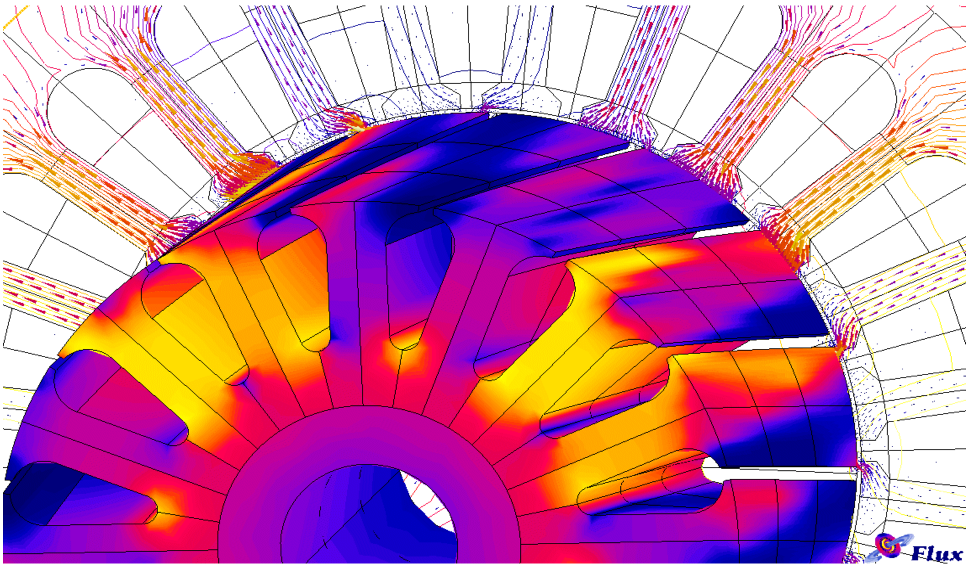
curve of the electromagnetic torque in function of the relative rotor – stator position and the angle of the rotor slot twist

Display of the volume regions

In a SKEW application, during the geometric and physical construction, the Flux project is a 2D problem. The different regions of the studied device are modeled by surface regions. Starting with Flux V11.10, after solving, the postprocessing offers 3D results. Previously, it was not possible to visualize physical quantities related to the volume regions. Starting with the V11.2.0 Flux version, the user can visualize physical quantities on these regions.

Rapidity of the 3D geometry building

The 3D SKEW geometry is constructed while the user launches the solving process of a project. The time required for building the regions has been diminished in the version V11.2.0.



Continued on next page

Comparison V11.1 vs V11.2

For informational purposes only, a comparison of the solving time between the V11.1.0 and the V11.2.0 has been carried out on the first case of the Skew tutorial. It is the model of a no-load steady state operation of an induction motor,in the Steady State AC Magnetic (MH) application. The table below summarizes the results obtained :

Test condition	Time for solving	
	Flux V11.1.0	Flux V11.2.0
RAM = 16 Giga OS = Windows Seven 64B Memory no = 1785Mo No of core = 1/8 Solver = Mumps sequentiel	1 hour and 29 minutes	52 minutes

- « Stop / Resume a solving » management

During the solving of a transient problem, piloting by time or by position, or of a multiparametric problem, the user can stop at any moment the solving process and resume the calculations later.

This function already exists in the 2D and 3D applications it has been implemented for the Skew in the V11.2.0 Flux version.

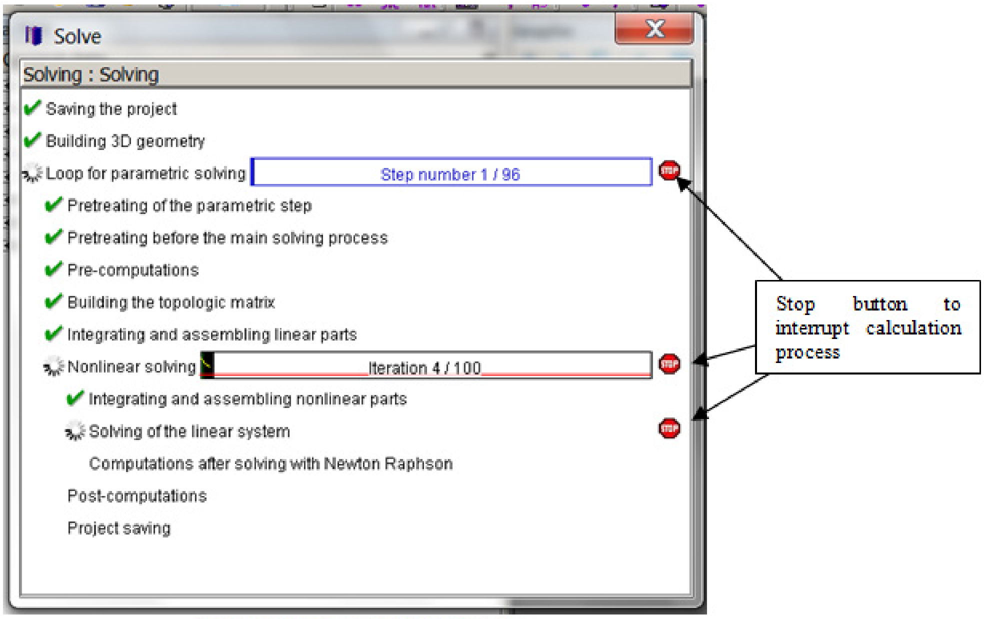


Figure 1 Windows resolution

To stop the solving process :	To restart the computation :
• click on the **STOP** buttons	• click on **Solving** menu • click on **Continue the solving process** • click on **Continue to solve a scenario**

6. V11.2 new features regarding Vibro-acoustic coupling

Introduction

The magneto-vibro-acoustic coupling facilitates the export of magnetic forces or harmonics according to the time variation of these forces, which are result of the transient magnetic Flux application. These results are then used with other software packages carrying out the vibro-acoustic portion of the computations, like LMS-VL or Nastran.
These analyses vary in the study of the function of the device (motor, actuator, ...).
The functionality exists in the 2D, Skewed and 3D models, with or without symmetries and periodicities.

Contents

This chapter contains the following topics:

6.1. The vibro-acoustic analysis overview

Definition

Any electromagnetic device (linear actuator, electrical motor etc.) comprised of a metallic part in motion surrounded by air is susceptible to generating vibrations and noise .
The calculation of the noise is made by taking into consideration the electromagnetic phenomena that generate the motion of the device.

Needs

The vibrations and the noise can be troublesome for the device itself, but also for the environment. It is then necessary to take into account such phenomena, starting with the conception phase, by carrying out a study of sensitivity with vibroacoustic parameters.

Solution brought by Flux

In order to offer the ability to carry out the computation of the acoustic noise, a dedicated context is integrated into Flux. It permits the user to cover all the stages up to the export of a **file of magnetic forces or of harmonics of forces**, which will then be used with the software for the vibro-acoustic study.
This context gives access to two types of computations, presented in the following table:

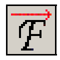
Computation	Exported quantity	Computation support	Computation method	Format of generated file	Associated software
Computation on regions contour	Magnetic forces	Flux entities	Derivative of energy – Maxwell tensor (method used with Flux)	.UNV	LMS-VL or others
Computation dedicated to Nastran coupling	Harmonics of forces	Mechanical meshing	Computation of the radial and tangent magnetic pressure in the airgap	.BULK	Nastran only*

*The Femap software is equally compatible with the vibro-acustic coupling.

6.2. Mechanical analysis context: about

Environment

In a project that consists of a **transient magnetic** (TM) **application**, the computation of magnetic force is accessible in the **Calculation** menu.

☞ **Computation → Open mechanical analysis context**	

Necessary condition

The context is accessible under the following conditions :
- Project defined as **transient magnetic** (MT) **application**
- Project representing a **time domain analysis** or a **multi-position analysis**

Remark 1: in this document we will always speak about «time». We should keep in mind that we could equally reason in position.

Remark 2 : the context is accessible for a parameter project

Valid domains

This context is accessible for the following Flux models :
- 2D plan
- Skewed
- 3D

In the axisymmetric 2D models the context is not available.

Main steps

In the table below the main steps to generate a file of forces are described.

Step	Action
1	Create the computation support
2	Compute forces
3	Check result by displaying arrows
4	Export forces

Support : definition

The support represents a mesh on which the computation of forces is carried out. The denser the mesh is, the more accurate the computation of forces will be.

Continued on next page

Computation : forces / harmonics of the forces

According to the chosen method of computation, there will be magnetic forces or harmonics of forces time variation exported. The user has the ability to choose to computation :

- Only on the forces
- On the forces and the harmonics

Advice: For all the chosen computation methods, the second choice is strongly recommendable – to **compute the forces and the harmonics**; the computation is very rapid, and the results are graphically visible in order to verify this computation.

6.3. Tools of geometric reconstruction and temporal duplication

Introduction

For needs of compatibility with the vibro-acoustic software, or in order to reduce the working time, geometric reconstruction and temporal duplication tools are available.

3D geometric reconstruction (automatic)

The vibro-acoustic software is 3D based. Consequently, whatever the initial Flux geometry may be, all the files exported by Flux represent results in 3D. The geometric reconstructions run by the software are the following :

- A 2D project (or Skewed) is reconstructed in 3D
- A model with symmetries and periodicities is reconstructed for the complete device

Temporal duplication: presentation

In the computation box of forces, the user can choose the option of a temporal duplication. This consists in transmitting the computed forces over a defined time interval, in order to automatically obtain the values across a mechanical period.

The advantages are :

- The computation of the forces is faster
- There is no need to solve the project on an entire mechanical period

Temporal duplication: in practice

The user has the option to choose a time interval corresponding to one portion 1/n* of the mechanical period.
**n : an <u>integer</u> number, worth minimum 2p (p : number of pairs of poles) for the electrical motors*

Temporal duplication: warnings

- Warning 1: the initial and final point of the time interval must merge. For example, if the period is of 0.06 seconds we will take :
    - An interval of 0 to 0.06 seconds without duplication
    - An interval of 0 to 0.015 seconds for a duplication of 4 times

Warning 2 : one should not take as a first step in the calculation of forces the first step of calculation of the solving scenario for reasons of numerical initializations

Continued on next page

Temporal duplication: verification

The accuracy of the results will depend on the chosen time interval. If the fraction 1/n of the period is not defined with an entire n, then the results will be distorted.

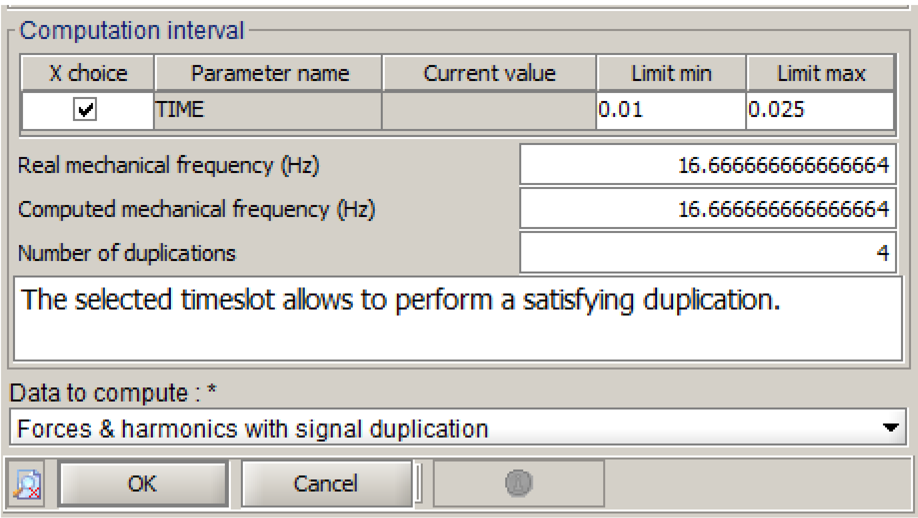
In the computation box, once the time interval is introduced and the choice on the time duplication has been carried out, the user has the option to verify the accuracy of his computation (before launching it) by comparing :

- The real mechanical frequency
- The computed mechanical frequency (using the chosen time interval)

The number of duplications, which is always a whole number, is also given. A message indicates the possibility or not for the software to apply the temporal duplication.

Example

The image below represents the part on the time interval of computation in the computation box. In this case, the machine rotates by 1,000 rpm, therefore the mechanical period is of 0.06 seconds. The machine has two pairs of poles, therefore it can choose a time interval at least equal to 1/2p=1/4 of the period (choice made in this case).



6.4. Computation dedicated to NASTRAN coupling

Introduction

This section describes all the aspects and steps involved in to generate the file for harmonics in the time variation of forces.

Contents

This section contains the following topics:

6.4.1. Nastran – utilization case, objective and method of computation

Case of nominal utilization

In the case of the vibro-acustic coupling with Nastran, the computation of the forces and of the harmonics of the forces is only dedicated to the **motor** applications.
The case of nominal utilization is a **motor operation with constant speed**.
In this case, the time duplication can be carried out.

Case of non nominal utilization

It is not forbidden to study a motor with variable speed or an asynchronous motor. In such cases, since the mechanical period is not known, the software cannot verify the validity of the returned time interval, nor to make a temporal duplication. Therefore, the user must choose a pertinent period for his computation.

Objective of the Nastran computation

The objective of the computation of forces for Nastran is to **export a file .BULK of the harmonics of the forces**, in order to use it in the vibro-acoustic software.

Computation method

The exported harmonics are calculated starting from the magnetic forces that are represented on a mechanical period.
The forces are themselves calculated starting from the magnetic radial and tangent pressures within the airgap. The forces densities are presented below.

Normal component : $\frac{1}{2} \times \frac{1}{\mu 0} \times Bn^2 - \frac{1}{2} \times \mu 0 \times Ht^2$

Tangential component : $Bn \times Ht$

Imported support and final computation support

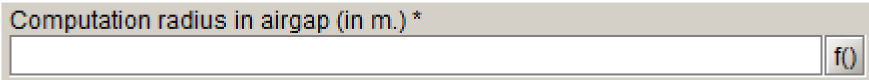
In the case of the Nastran coupling, the user must import a support representing the 3D mechanical mesh (file **.BULK**). This support must be at the interface of two regions of different permeabilities. For example: at the interface between the stator and the airgap, or between the rotor and the airgap.
The calculation is carried out at the interior of the airgap and then it is projected on the imported support. The radius within the airgap is asked for in the force computation box (see the paragraph below Nastran – Computation radius in airgap and slots opening angle).

Remark: The Femap software is also compatible with the Nastran coupling of FLUX. This software generates the same mesh in the file .DAT. It is sufficient to rename it as .BULK in order to import it.

6.4.2. Nastran – Computation radius in airgap and slots opening angle

Computation radius in airgap: definition

The computation radius in airgap defines the support where the computation of forces will be carried out, before the projection on the imported mechanical support.
The window to define it is shown below.

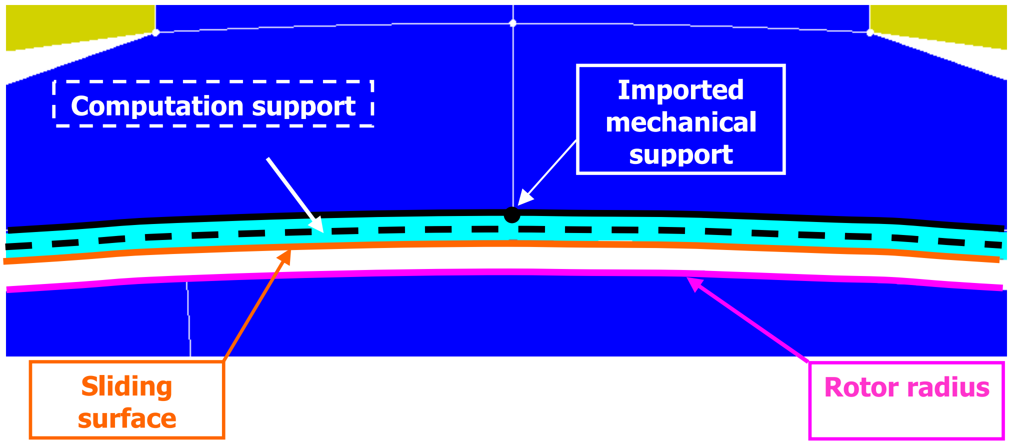


Computation radius in airgap: piece of advice

It is advisable to take a radius situated at a quarter of the airgap thickness on the side of the stator if the imported support is on the stator or on the side of the rotor if the imported support is on the rotor.

Example: support between stator and air

In the figure below an imported support on the side of the stator is represented, with the computation radius situated at a quarter of the airgap on the side of the stator.

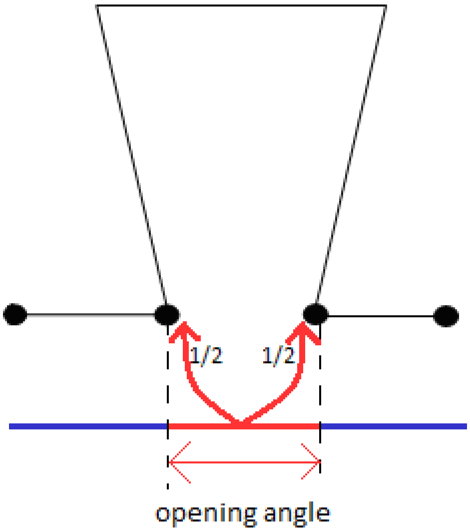


Continued on next page

Slots opening angle: definition

The slots opening angle is a parameter that must be defined by the user (see the figure below).
Introducing its value permits the software to take into consideration the values of the computed forces on a portion of the computation support so that they should be projected in equal parts on one side and on the other. The associated window presented below is found in the computation box.

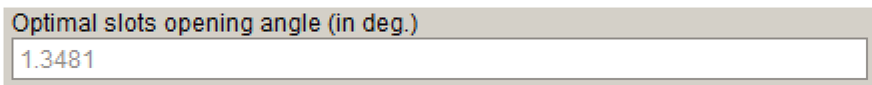
Example : Slots opening angle for a computation on the stator side



If the user does not wish to take into consideration this portion for the computation support, it is enough to choose a null value of the slots opening angle.

Slots opening angle: optimal value

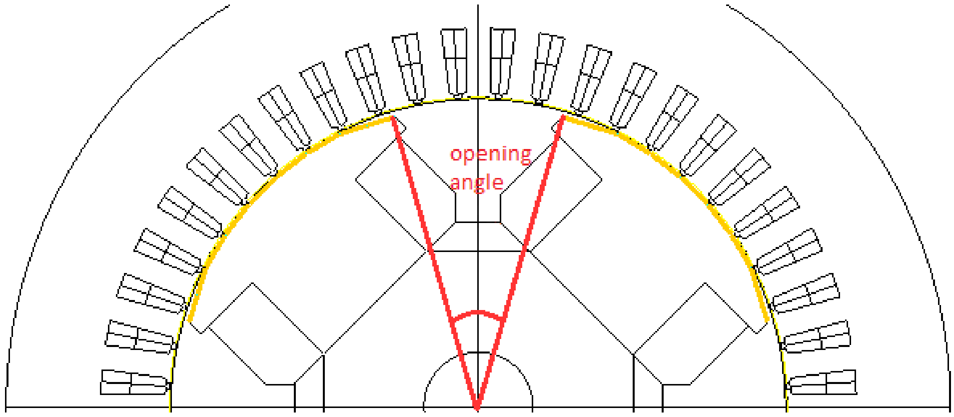
Once the computation with the slots opening angle is introduced, the optimal opening angle is calculated by the software and displayed in the computation box (see the image below).
This is useful for verifying the entered value of the slots opening angle, especially as it happens that the mechanical mesh might not be exactly merged with the geometry (as the geometric accuracy is not the same).



Continued on next page

Slots opening angle: computation on the rotor

If the computation of the forces is carried out on the rotor side, the choice of opening angle is often not easy according to the geometric complexity of the rotor.
The user is free to make his choice while being clear as to the imported mechanical support.
Example :



6.4.3. Nastran - Step 1 : Creation of the computation support

Introduction

In this section, the approach for creating the Nastran support is presented.

☞ **Computation support → New**

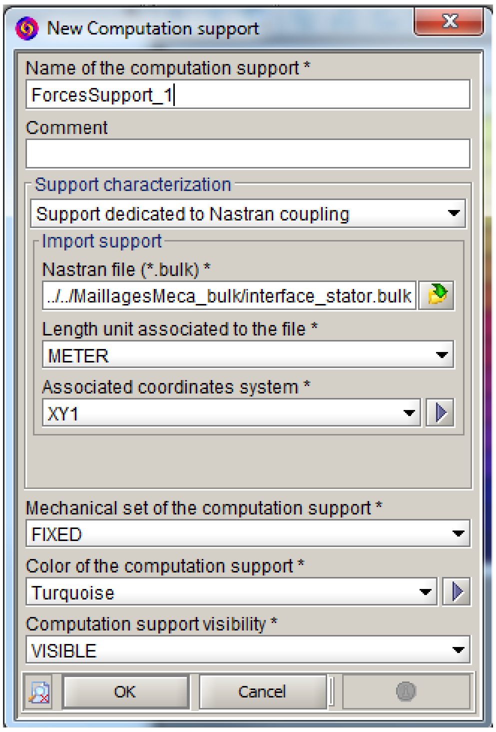
Creation of the support

The steps in creating the Nastran support are described below. Images of the creation box of the support are also available.

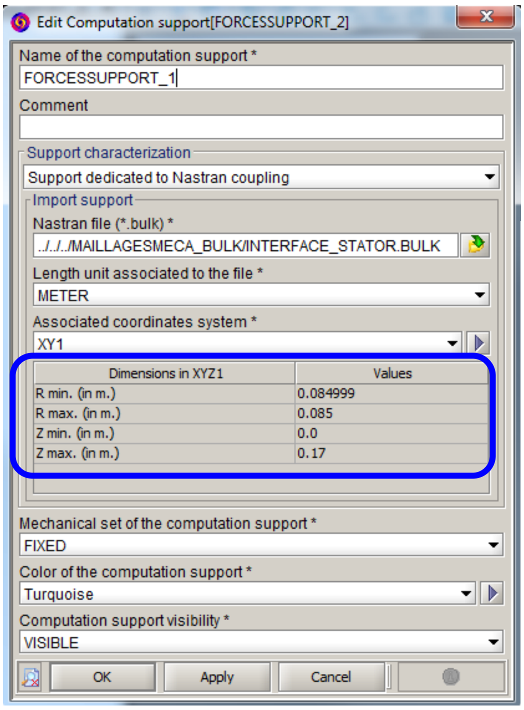
Step	Action
1	Choose the name of the support (+ comment)
2	Choose the method **Support dedicated to Nastran coupling**
3	Import the file .BULK representing the mechanical meshing
4	Choose the measurement unit to be taken into consideration at the import
5	Choose the coordinate system* associated to the imported support (see the following sections)
6	Choose the mechanical set of the imported support : • Fixed if the support is on the stator • Mobile if the support is on the rotor
7	OK
8	Visually verify the placement of the imported support with respect to the geometry. The visualization in 3D is possible irrespective of the project dimension.
9	Re-open the created support box
10	Verify the dimensions Rmin, Rmax, Zmin, Zmax of the imported support knowing that : • Rmin, Rmax : minimal/maximal radius among all the nodes in the global coordinate system XYZ1 of FLUX • Zmin, Zmax : coordinate on Z minimal/maximal among all the nodes in the global coordinate system XYZ1 of FLUX

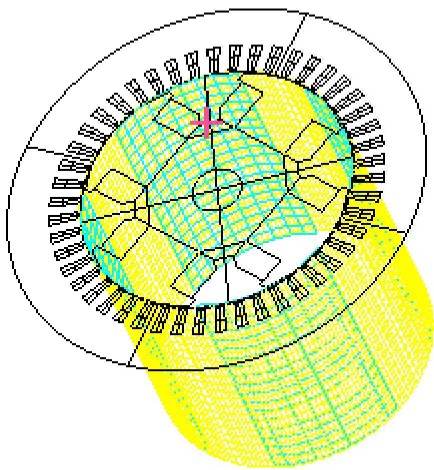
Continued on next page

Step 1 to 7 :



Step 8 to 10 :





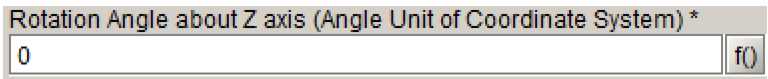
Continued on next page

***Coordinate system of the support: Presentation**

Often with mechanical software, the coordinate system in which the motor is defined is not a coordinate system of its own, but the one of the global device (car, …).
With FLUX, it is therefore necessary to make the imported support correspond to the geometry. The geometry must have the **main axis merging with the axis Z of the XYZ1 coordinate system.**

***Coordinate system of the support: In practice (advice)**

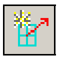
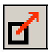
In practice, it is very advisable to create two coordinate systems. The approach required is presented in the following table :

Step	Action
1	Create a first coordinate system NASTRAN_1 defined in the XYZ1 coordinate system of FLUX
2	Create a second coordinate system NASTRAN_2 defined in the NASTRAN_1 coordinate system
3	Apply in NASTRAN_1 according to the needs : • The translation according to x/y/z (in XYZ1) • The rotation around x/y/z (in XYZ1) This permits merging the position of the imported support with that of the FLUX geometry
4	Apply in NASTRAN_2 the rotation of the machine around its main axis, that is merged with the axis Z of NASTRAN_1 : • Modify the following window 

6.4.4. Nastran – Step 2: Creation of the force computation and export of the harmonics

Introduction

In this section, the method of creation of a computation support and the export of the harmonics are presented.

☞	Force computation → New	
☞	Force computation → Export Force for Nastran	

Reminder : Objective of the Nastran computation

The objective of the computation of the forces for Nastran is to **export a file .BULK of the harmonics of the forces**, in order to use it in the vibro-acoustic software.

Continued on next page

Creation of the force computation

The steps to create the Nastran force computation are described below.
Images of the creation box are also available.

Step	Action
1	Choose the name of the force computation (+comment)
2	Choose the method **Computation dedicated to NASTRAN coupling**
3	Choose the support created beforehand
4	Define the computation radius in airgap on ¼ distance of the airgap thickness, stator/rotor side, according to where the imported support is (see Nastran – Computation radius in airgap and slots opening angle)
5	Choose the slots opening angle (See Nastran – Computation radius in airgap and slots opening angle)
6	Choose the interval of computation according to the choice of applying or not a temporal duplication in the field **Data to be computed** (see Tools of geometric reconstruction and temporal duplication). Reminders : • The initial step and the final step are repeated • Do not take the first step of the scenario of the solving process as the first step of computation

7	Choose the data to be computed. You must choose :
	• Application or not of a temporal duplication (choice connected to the step 6)
	• The choice to calculate :
	  • Only the forces
	  • Or the forces with their harmonics
	It is strongly recommended to calculate both forces and harmonics, as the calculation of the harmonics is rapid

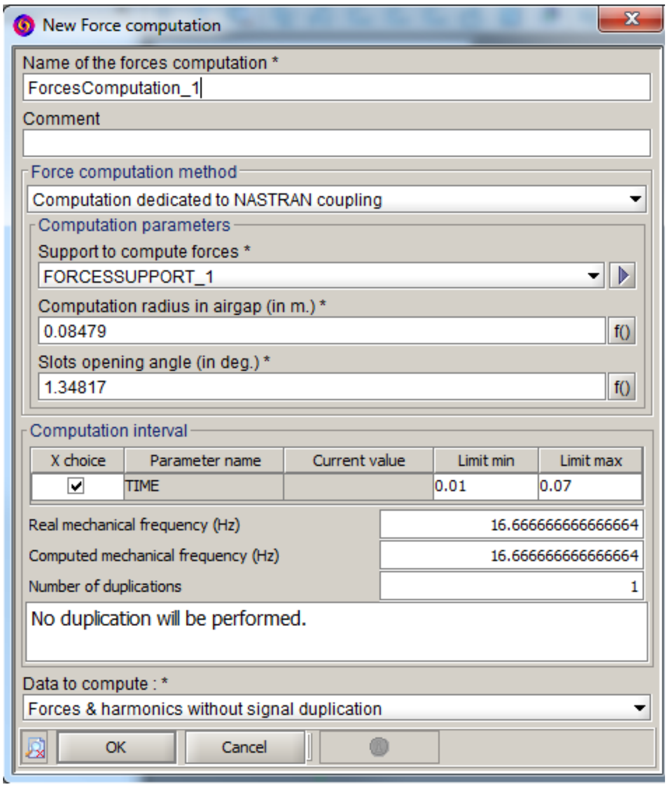
8	Verify if the time interval is in good correlation with : • The real mechanical frequency • The computed mechanical frequency (based on the computed interval) • The number of duplications (see Tools of geometric reconstruction and temporal duplication)
9	Validate by OK

Continued on next page

10	Re-open the computation box and verify : • The optimal opening angle (See Nastran – Computation radius in airgap and slots opening angle) • the quality of the radial and tangential projection (See Nastran – Projection quality rate)
11	Visualize the arrows of the forces or of the harmonics (see Display arrows and animation)

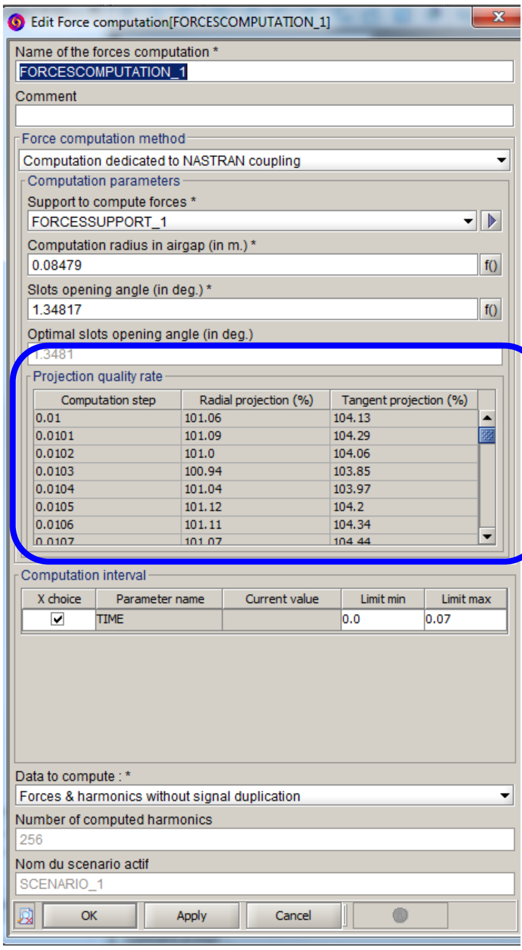
12	Export the harmonic's specifics :
	• the name of the computation
	• the number of harmonics to export
	    • "nothing" to export all the harmonics
	    • n first harmonics (n to be specified)
	• Force unit
	• Name of the file of .BULK type

Step 1 à 9 :

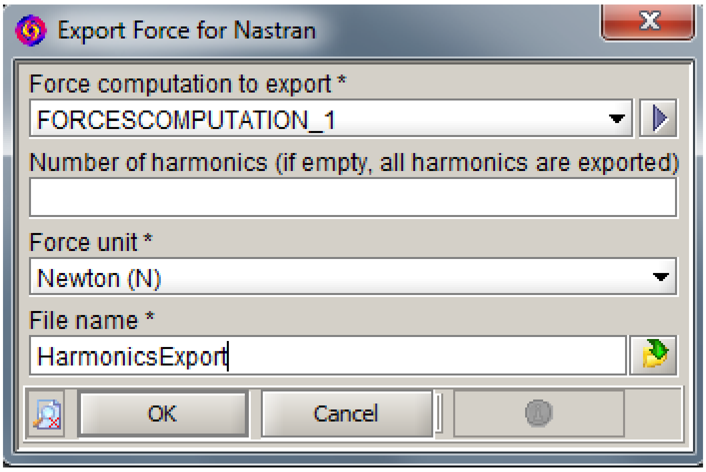


Continued on next page

Step 10 à 12 :



Step 13 :



6.4.5. Nastran – Projection quality rate

Projection quality rate : interest	The forces projection quality factor will help to verify the good progress of forces computation and projection on the mechanical mesh.
Projection quality rate : Influence of application	According to the application dimension (2D, 2.5D, 3D), the projection and control methods are different. The forces projection quality factor values (radial and tangent) could vary depending on the concerned applications.
Projection quality rate : In practice	The projection quality is not an absolute reference but a guide number to avoid aberrant results. In practice, we will not try to reach an exact 100% quality. It should be noted that the projection quality control method is more adapted to radial forces. So we will verify as a priority the projection quality of those components. The projection quality for tangent forces is given for information purposes only. For example, we can consider that the radial projection quality is generally higher than 90%, whereas the tangent projection quality can be easily deteriorated to reach sometimes values around 50% for 2.5D applications especially.

6.5. Computation on regions contour (e.g.: LMS-VL)

Introduction

This section describes all the aspects and steps permitting the file of forces generation for computation on regions contour, notably compatible with the **LMS-Virtual Lab** software.

Contents

This section contains the following topics:

6.5.1. Computation on regions contour - Use case, objective and computation method

Case of nominal utilization

The initial context is devoted to **motors operating at constant speed**. In this case, the temporal reconstitution is dealt with and one can propose a means of verification by the frequency of the fundamental harmonic.

Case of non nominal utilization

Nevertheless, there is nothing to prevent the study of other configurations, if the user masters the returned data. Indeed, with the temporal reconstitution not being dealt with, the user has the task to choose the period which is pertinent to his study. Here are some examples :

- Rotating actuator with variable speed (mechanical set with coupled charge)
- Linear actuator
- Fixed device

Objective of computation on regions outline

The objective of computation of the forces on the regions outline is to **export a file .UNV of forces**, in order to use it with the software for vibro-acoustic analysis.

Computation method

The computation method is already used with FLUX, it concerns the energy derivative by using the Maxwell tensor.

Computation support

The computation method on the region outline is applied on a support defined by the geometric entities of FLUX :

- Lines in 2D
- Faces in Skew and 3D

The support must necessarily be at the interface of two regions of different permeabilities (stator/air, or rotor/air), and it should not be on the sliding cylinder.

6.5.2. Computation on regions contour – Step 1 : Creation of the support

Introduction

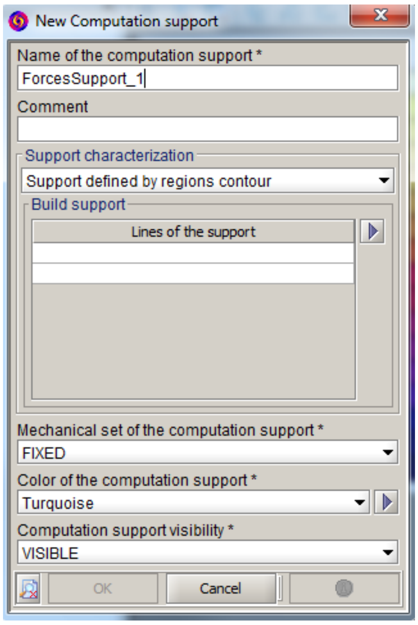
The method to create the support to carry out a computation on the region contour is described in this section.

☞ Computation support → New

Support creation

The steps in creating the computation support are described below. An image of the creation box for the support is also available.

Step	Action
1	Choose the name of the computation support (+ comment)
2	Choose the method **Support defined by regions contour**
3	Select the Flux geometric entities to create the computation support*: • Lines in 2D • Faces in Skew and 3D An automatic filtering exists, refusing the selection of non valid entities
4	Visually verify the computation support. The visualization in 3D is possible whatever the project dimension may be.
5	Choose the mechanical set of the computation support : • Fixed if the support is on the stator • Mobile if the support is on the rotor
6	Validate by OK



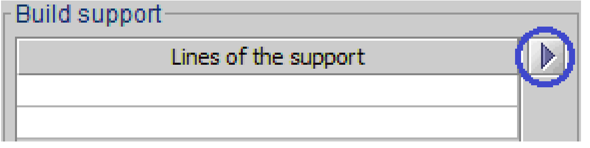
Continued on next page

***Tip on computation support selection**

Given the existence of an intelligent filter that does not select the invalid elements, one tip is to pass through a constraint region :

- Choose the stator region as the constraint if the computation support is on the stator side
- Choose the rotor region if the computation is on the rotor side.

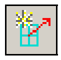
Only the geometric entities verifying the constraint of the method are then selected.

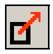


6.5.3. Computation on regions contour – Step 2 : Computation and export of the forces

Introduction

The method of computation of the forces (and harmonics) on the region contour is described in this section.

☞	Force computation → New	

☞	Force computation → Export Force for Virtuallab	

Reminder : Objective of the computation

The objective of the computation of forces for the calculation method on the regions outline is to **export a file .UNV of magnetic forces**, in order to use it with the software for vibro-acoustic analyses.

Continued on next page

Creation of force computation

The steps of the computation creation are described below. Images of the creation box of the computation are also available

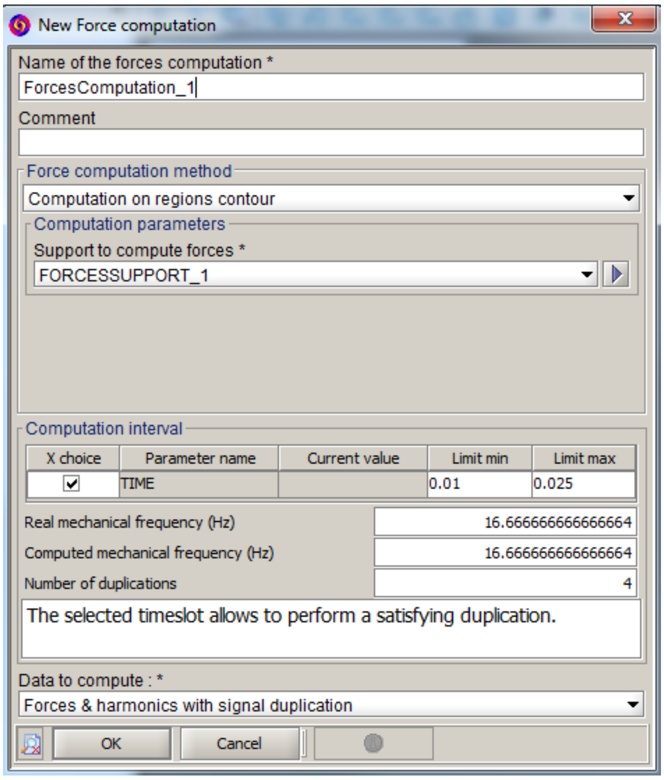
Step	Action
1	Choose the name of the computation (+comment)
2	Choose the computation method **Computation on regions contour**
3	Choose the computation support created beforehand
4	Choose the computation interval according to the choice of applying or not a temporal duplication in the field **Data to compute** (See Tools of geometric reconstruction and temporal duplication). Reminders : • the initial step and the final step are repeated • Do not consider the first step of the scenario as the first step of the computation

5	Choose the data to compute. It is about choosing :
	• The application, or not, of a temporal duplication (choice connected with stage 4)
	• The choice to compute :
	  • Only the forces
	  • Or the forces with their harmonics
	It is strongly advisable to calculate both the forces and harmonics, as the computation of the harmonics is rapid.

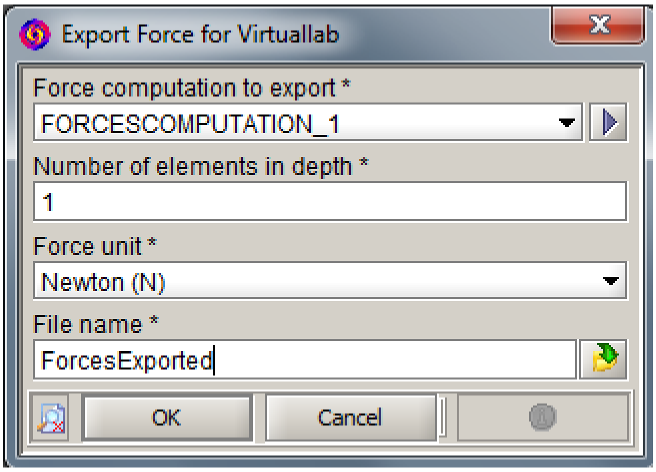
6	Verify if the time interval is in good correlation with : • The real fundamental frequency • The computed fundamental frequency based on the computation interval • The number of duplications See Tools of geometric reconstruction and temporal duplication .
7	Validate by OK
8	Visualize the arrows of the forces or of the harmonics (See Display arrows and animation)
9	Export the forces by specifying : • the name of the computation • In 2D, the number of elements in the depth* • Force unit • Name of the file .UNV

Continued on next page

Step 1 to 7:



Step 9:



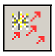
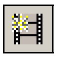
Number of elements in the depth

In 2D, the number of elements in the depth is required in order to project the results on the depth (by dividing the values by this number of elements in order to always have the same total sum).
In Skew, the number of elements is defined in the application of the project, and in 3D the forces are calculated on the nodes of the mesh.

6.6. Display arrows and animation

Presentation

This section presents the display of the arrows of the forces or of the harmonics, as well as the animations by using these arrows. These tools are means of verifying the computation carried out.

☞	**Graphic → Arrows …**	
☞	**Graphic → Animation**	

Display arrows

It is possible to display the following arrows :
- Forces of current step (not selected at the bottom left of the project)
- Harmonics of the continuous component of the forces
- Harmonics of the forces (in function of the phase)

Reminder: the arrows are visualized on the support in 3D whatever the dimension of the FLUX project.

Remark : the total sum of the forces is accessible in the FLUX console at the display of the arrows

Creation of an animation: Warnings

Starting from the previously created arrows it is possible to generate animations.
Here are some warnings :
- You must choose carefully the time interval of the animation so that it should not be included in the time interval of computation associated to the chosen arrows
- The animation gives access to the arrows of the harmonics but as they do not depend on time, the result will produce a fixed image

7. V11.2 new features regarding Material Manager

Introduction

This chapter deals with the new material manager, a tool dedicated to materials.

Context

In Flux software, the materials are managed:
- directly in the Flux project
- or by means of the material manager

This chapter deals with the management of materials via the material manager.

Contents

This chapter contains the following topics :

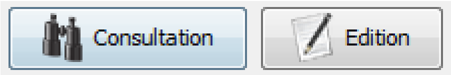
7.1. Material manager: about

Introduction

The material manager is an interface facilitating materials management in a database. It comprises two working contexts:

- Editing of databases: it permits the user to add, modify, import or delete materials and databases
- Consultation of the databases: it allows the user to see and compare the information on materials and to compare the curves.

The context choice is done by the following buttons :



Mono / multi databases

The editor context operates in « mono-database » and « mono-material » mode, i.e. the user cannot select more than one working database and open more than one material at a time.
The consultation context, however, works in « multi-databases » and « multi-materials » mode, making it possible to compare materials belonging to several databases.

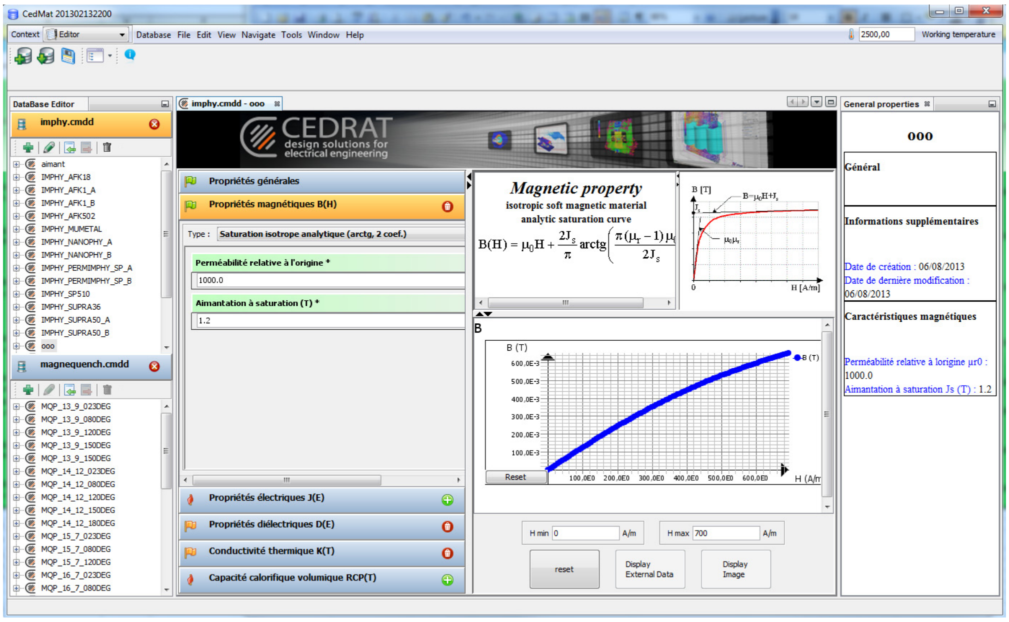
Access / installation

The material manager is installed with Flux and it is accessible via the supervisor by clicking on the button:

Environment

A welcome window is available and explains the different components of the application. The icon to display or mask this window is :

Continued on next page


CedMat 201302132200
Context Editor Database File Edit View Navigate Tools Window Help
2500,00 Working temperature
DataBase Editor
imphy.cmdd
magnequench.cmdd
imphy.cmdd - ooo
General properties
Propriétés générales
Propriétés magnétiques B(H)
Type : Saturation isotrope analytique (arctg, 2 coef.)
Perméabilité relative à l'origine *
1000.0
Aimantation à saturation (T) *
1.2
Propriétés électriques J(E)
Propriétés diélectriques D(E)
Conductivité thermique K(T)
Capacité calorifique volumique RCP(T)
Magnetic property
isotropic soft magnetic material
analytic saturation curve
H min 0 A/m
H max 700 A/m
reset
Display External Data
Display Image
ooo
Général
Informations supplémentaires
Date de création : 06/08/2013
Date de dernière modification : 06/08/2013
Caractéristiques magnétiques
Perméabilité relative à lorigine µr0 : 1000.0
Aimantation à saturation Js (T) : 1.2

7.2. Material database

Introduction

This chapter presents the definition of a database in the material manager and the existing database types.

Contents

This section contains the following topics :

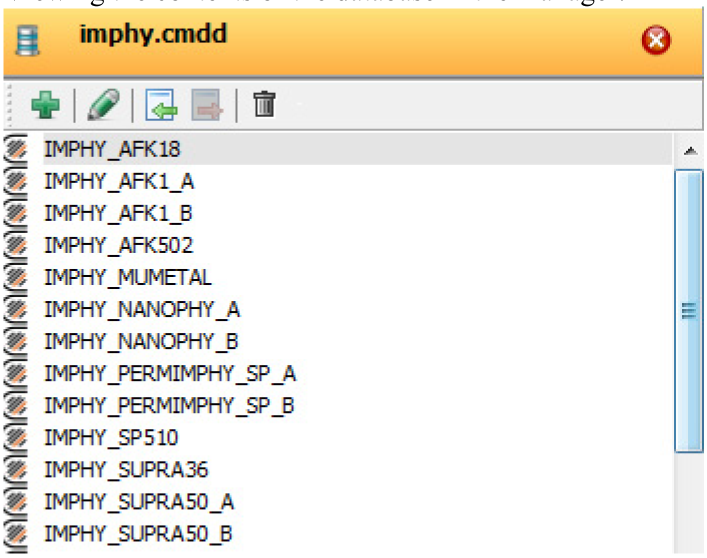
7.2.1. Material database: presentation

Database: definition

The materials database is a file comprised of a list of materials which includes: general data, physical properties, models and characteristics depending on the models. A material database is a file, named according to the following naming convention: DatabaseName.CMDD. The contents of a materials database is visible in the materials manager under the following form:

File of the database :

Viewing the contents of the database in the manager:



Database: settings

A database is characterized by:
- A name
- A path

Cedrat database and User databases

There are two types of materials databases:
- The **Cedrat material database** is supplied with the software and it is not modifiable
- The **user material databases**, are databases created by the user (DatabaseName.CMDD), and are therefore modifiable

7.2.2. Cedrat database

Introduction

The Cedrat database includes materials of different types originating from several manufacturers. It is not modifiable by the user. But it is possible to copy materials from this database to user database.
The icon to display or hide the Cedrat database is :

Naming norms

There are several nomenclature norms for materials (for example: AISI, ASTM, JIS, IEC).
In the materials manager, the name of a material supplied by Cedrat contains :

- Manufacturer name
- Material name chosen by the manufacturer
- The associated frequency (if available)

For example, the magnetic material M220-50A, manufactured by Imphy, characterized at the frequency 50 Hz. will be named Imphy_M220-50A_50Hz.
The names in other norms are referenced as well as possible in the “description” area of the « General » panel.

Manufacturers

The Cedrat database is comprised of materials coming from various materials manufacturers. For example:

- Imphy – Aperam
- Magnequench
- Thyssenkyupp
- China steel
- Cogent
- Protolam

7.3. Materials

Introduction

The materials used to define the physical properties of different regions in a Flux problem can be stored in a materials database.

Structure (definition)

In a materials database a material is defined by:
- **General data** on the material
- One or more **physical properties** (electrical, magnetic or thermal), themselves characterized by **models** represented by curves

General data

The general data for the materials are presented in the table below.

Data	Contents / Remarks
Name	
Comments	Limited number of characters
Family	It allows to sort materials in the tree by family
Chemical composition	
Manufacturer	It allows to sort materials in the tree by manufacturer
Reference	
Mass density (Kg/m3)	
Price ($/Kg)	
Author	
Lamination data	In the case of soft magnetic materials, it is possible to specify if it is in the form of laminations or a solid body
	In the case of laminations, the following characteristics can be entered: thickness, specific losses at 1T and 1.5T for a given frequency, fill factor
Description	Text field without limitation of characters

Continued on next page

Physical properties

The **physical properties** that permit the characterization of the materials are listed in the table below.

Physical properties	Law of behaviour
Magnetic	Loi B(H)
Electric	Loi J(E)
Dielectric	Loi D(E)
Thermal conductivity	Loi k(T)
Heat capacity	Loi $\rho C_P(T)$

A material can have an array of properties, independent of the physical application in which it will be used.

The « dependences » between the physical properties and the physical applications are presented in the chapter « Materials: software aspects » (online help).

Models

The available **models** for the various physical properties are presented in a recap chart at the end of this chapter (see List of models for each properties). A more detailed content on each model exists in the chapter Materials: principles.

The presented models in Flux and in the materials manager are compatible.

7.4. List of models for each properties

Models

There are numerous models which describe the physical properties of the materials. These models are presented in the chapter Materials: principles. The material manager models, globally corresponding to those of Flux, are presented in the tables below.

	B(H) for soft materials	Flux	Manager
Linear	Linear isotropic	✓	✓
	Linear anisotropic		
Linear complex	Linear isotropic complex		
	Linear anisotropic complex		
No linear	Isotropic analytic saturation (arctg, 2 coef)		
	Anisotropic analytic saturation (arctg, 2 coef)		
	Isotropic analytic saturation + knee adjustment (arctg, 3 coef)		
	Anisotropic analytic saturation + knee adjustment (arctg, 3 coef)		
	Isotropic spline saturation		
	Anisotropic spline saturation		
	Rayleigh isotropic : parabola + straight line		

	B(H) for hard materials	Flux	Manager
Linear	Linear magnet described by the Br module	✓	✓
	Linear magnet described by cartesian vector Br	✓	✗
	Linear magnet described by cylindrical vector Br		
	Linear magnet described by spherical vector Br		
No linear	Nonlinear magnet described by Hc and Br module	✓	✓
	Nonlinear magnet described by Hc and Br module + knee adjustment		
	Nonlinear magnet described by spline		

	B(H, T)	Flux	Manager
Linear	Linear isotropic * exponential function of T	✓	✓
	Linear isotropic, tabulated function of T		
No linear	Isotropic analytic saturation * exponential function of T		
	Isotropic analytic saturation + knee adjustment * exponential function of T		
	Isotropic analytic saturation , tabulated function of T		

Continued on next page

<table>
<tr><th></th><th>D(E)</th><th>Flux</th><th>Manager</th></tr>
<tr><td rowspan="2">Linear</td><td>Linear isotropic</td><td rowspan="4">✓</td><td rowspan="4">✓</td></tr>
<tr><td>Linear anisotropic</td></tr>
<tr><td rowspan="2">Linear complex</td><td>Linear isotropic with losses (tg δ)</td></tr>
<tr><td>Linear anisotropic with losses (tg δ)</td></tr>
</table>

<table>
<tr><th>J(E)</th><th>Flux</th><th>Manager</th></tr>
<tr><td>Insulator</td><td rowspan="3">✓</td><td rowspan="3">✓</td></tr>
<tr><td>Isotropic resistivity</td></tr>
<tr><td>Anisotropic resistivity</td></tr>
<tr><td>Isotropic superconductivity</td><td>✓</td><td>✗</td></tr>
</table>

<table>
<tr><th>J(E, T)</th><th>Flux</th><th>Manager</th></tr>
<tr><td>Isotropic resistivity, linear function of T</td><td rowspan="5">✓</td><td rowspan="5">✓</td></tr>
<tr><td>Anisotropic resistivity, linear functions of T</td></tr>
<tr><td>Isotropic resistivity, exponential function of T</td></tr>
<tr><td>Anisotropic resistivity, exponential functions of T</td></tr>
<tr><td>Isotropic resistivity, tabulated function of T</td></tr>
</table>

<table>
<tr><th>k(T)</th><th>Flux</th><th>Manager</th></tr>
<tr><td>Isotropic conductivity, constant (independent of T)</td><td rowspan="7">✓</td><td rowspan="7">✓</td></tr>
<tr><td>Anisotropic conductivity, constant (independent of T)</td></tr>
<tr><td>Isotropic conductivity, linear function of T</td></tr>
<tr><td>Anisotropic conductivity, , linear functions of T</td></tr>
<tr><td>Isotropic conductivity, exponential function of T</td></tr>
<tr><td>Anisotropic conductivity, , exponential functions of T</td></tr>
<tr><td>Isotropic conductivity, tabulated function of T</td></tr>
</table>

<table>
<tr><th>$\rho C_P(T)$</th><th>Flux</th><th>Manager</th></tr>
<tr><td>Heat capacity, constant (independent of T)</td><td rowspan="6">✓</td><td rowspan="6">✓</td></tr>
<tr><td>Heat capacity, linear function of T</td></tr>
<tr><td>Heat capacity, exponential function of T</td></tr>
<tr><td>Heat capacity, Gaussian function of T + constant</td></tr>
<tr><td>Heat capacity, Gaussian function of T + exponential</td></tr>
<tr><td>Heat capacity, tabulated function of T</td></tr>
</table>

7.5. « Edition of databases » context

Introduction

The edition context allows the user to add, modify, import or delete materials and databases. It works in « mono-database » and « mono-material » mode: one material is opened in one working database.
It is available via the following button:

Contents

This section contains the following topics :

7.5.1. Main functionalities

Manage databases

The first step in this context is to choose a database. The actions for database management are presented in the table below.

Command	Access	Function
Create a database		Create a database with a name and a saving location
Load a database		Load an existing database in the materials manager in order to manage its materials
Unload a database		Unload a database when there is no more need to manage its materials

Manage materials

Once the database is chosen, it is then possible to manage its materials. The materials management actions are presented in the table below.

Command	Access	Function
Create material		Create a new material
Delete a material		Delete an existing material
Modify a material	or double-click	Modify the data of a material
Copy a material	Contextual menu: Copy and paste on a sorting directory*	Copy a material into another database
Duplicate a material	Contextual menu: Copy and paste on a sorting directory*	Duplicate a material within the same database
Import a material		Import a python file comprised of the creation commands for the materials
Python export of a material	*	Export the python commands for materials creation into a python file

* Those actions are also possible with a drag and drop with the mouse

Continued on next page

Manage materials data

Once the material is created or opened, it is then possible to manage its general data and properties. The materials data management actions are presented in the table below.

Command	Access	Function
Associate general data at the material	General	Associate general data (7.3)
Associate properties to the material	Access to the magnetic property: B(H)	It is possible to associate magnetic, electric, dielectric and thermal properties to the material. (7.3)
	Access to the model choice: Type : Saturation isotrope spline	Associate an analytical or a numerical model to a property (7.3)
Copy or replace a property	Contextual menu: Copy and paste on the material*	Copy a material property into another material (add or replace)

* Those actions are also possible with a drag and drop with the mouse

General commands

The general management commands are presented in the table below.

Command	Access	Function
Save		Saves the databases which were modified
Exit		Leaves the application
Help	"Help" menu: help	Opens the Flux html help containing the material manager help

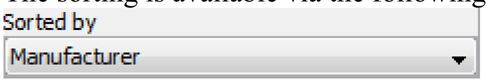
7.5.2. Useful functionalities

Sorting materials

It is possible to sort out the materials lists in their database by:

- Manufacturer
- Family
- Property
- Alphabetical order

The sorting is available via the following component :



Model creation help

A help window for model creation is available through the following icon:

Summary

A summary is automatically built for each material from the completed data. It contains the following parts:

- General: the general data
- Additional information: the date of creation and of modification
- Magnetic characteristics: the name of the model and **the physical parameters recovered or evaluated starting from the model** (it allows to verify the model)
- Electric/thermal characteristics : the name of the model

The icon to display or hide the summary window is :

Curve

Once the material model is completed, the associated curve is ploted.

Import of materials created in Flux

In order to recover the materials created in Flux in the materials manager, the steps to follow are:

- In Flux, export the materials to a python file
- In the materials manager, import the python file into the chosen database

Continued on next page

Import of materials in .DAT files (cslmat)

In order to store materials of .DAT files (old format) into .CMDD database (new format), the first step is to import them in a Flux project*. Then the two steps presented above allow to make the import.

* The Flux command which allows to import the .DAT materials is :

7.5.3. B(H) model creation tools

External data B(H) : Interest

It is possible to load B(H) data. Those data are stored in the application and are visible in the curve graphic independently of the model. It is useful for the model creation:

- Spline model : for the construction of smoothed or non smoothed spline
- Analytic model : for the manual matching

External data B(H) : In practice

The data are loaded by one of the two methods :

- Import a txt or csv file
- Paste data copied from an excel file

The data format corresponds to two columns values. Most of usual separators work (dot, comma, space…).

Spline wizard

To create a spline model, except the standard mode of entering values one by one, a wizard is available with the following functionalities :

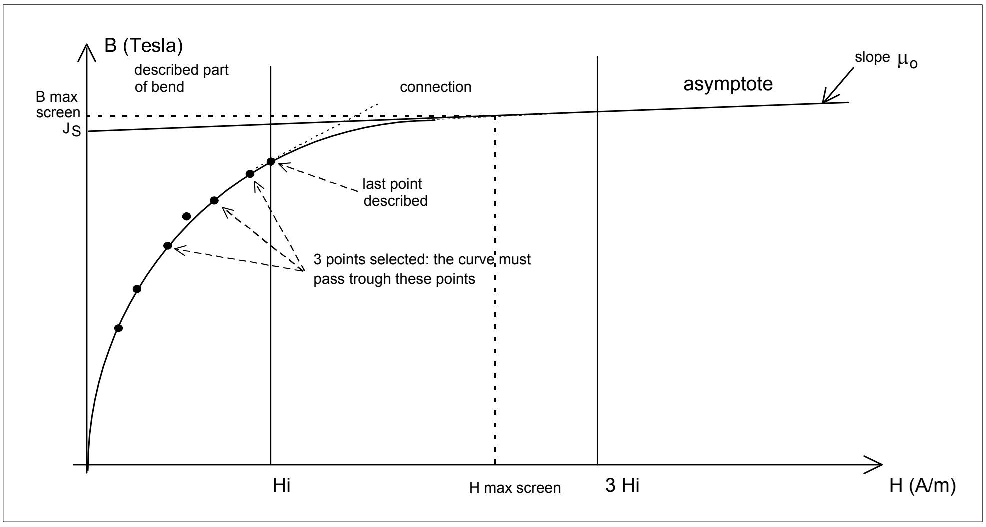
- Enter the external data in the wizard if it is not already done in the way presented in the previous block
- Choose if a smoothing have to be applied or not on initial values

Smoothed spline : model explanation

The spline saturation model is build in three parts:

- The first part is a homographic function passing through the origin and describing the bend. The function is defined by the pair of values B,H supplied by the user and ends at the last pair of values supplied
- The second part is a connecting homographic function, tangent to the first and last parts of the curve
- The third part is a straight line of slope μ_0, whose ordinate at the origin is the saturated magnetization $\mathbf{J_s}$.

Continued on next page



Smoothed spline : data to enter

As a reminder, the non smoothed spline construction needs the B, H values of all the first magnetization curve, until the saturation (the model goes through all data points).

The smoothed spline needs :

- Points describing the knee. The number can vary from 3 to 200 points but in general 5 to 8 representative points are enough
- Three imposed smoothing points on the experimental curve. The spline saturation curve will pass through these three points.
- Saturation magnetization Js (in Tesla)

7.6. “Consultation of databases” context

Introduction

The consultation context allows the user to see and compare the information on materials and to compare the curves. It works in multi-databases and multi-mateials mode.

It is available via the following button:

Contents

This section contains the following topics :

7.6.1. Consult and compare

Presentation

In the consultation context, it is possible to select one or several materials of one or several databases. It allows to consult and compare the following data:

- Superimposed curves
- Comparative table

Comparative table

The comparative table allows to have the summaries contents of all the selected materials.
It is especially interesting to compare the physical parameters which are recovered or computed from the model.

For example : for the spline model the following parameters are evaluated :

- Saturation magnetization J_s
- Initial relative permeability μ_{r0}
- Mean evaluated permeability μ_{moy}

Superimposed curves

In consultation it is possible de superimpose the curves on the same graphic. An easy visual comparison can be done.

7.6.2. Tools of displaying for curves

Tools of displaying for curves

In the two contexts, there are tools available for displaying curves. These tools are presented in the following table.

Note: for this version, it is not possible to display curves apart from the magnetic property.

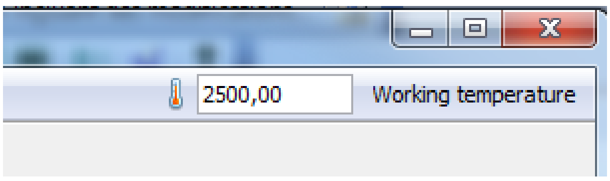
Display tools	Function
Legend	
Scale	Choice between: • The automatic mode • The user mode, where H_{min}, H_{max} are provided by the user
Discretization	Number of curve discretization points *
Standard tools	• Zoom • Cursor in order to display the values (permits to calculate the distance dx and dy between two points) • Cursor to move the curve • Reset (the three previous parameters recover default state)
External data visualization (Edition context)	• Choice of displaying external data points or not

* the displayed curve is smooth but in reality it is the result of computed discretization points.

Particular case: anisotropic or depending of T

In the case of an anisotropic material or a property depending on the temperature, the material has not only one characteristic curve:

- Anisotropic material: the curves along the axes x, y and z are simultaneously displayed on the same graphic
- Material with property depending on the temperature: the curve is displayed at the working temperature globally used in the application (see image below)



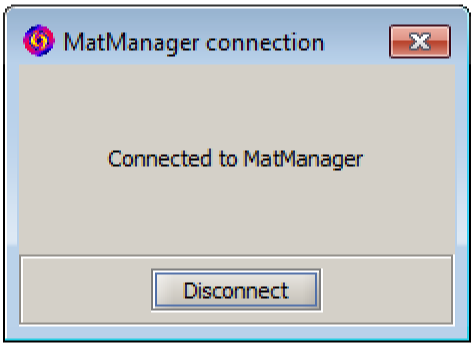
7.7. Import materials in Flux project

Presentation

In reality, it is an import launched by a FLUX project. The materials import from the material manager into a Flux project is the main way to import materials in FLUX. The different steps to follow are presented in this paragraph.

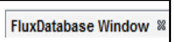
Technical principle of Flux import

The data transfer from the material manager to Flux is done by a temporary connexion between the two applications, linked to a modal window of Flux import. When the import command is launched, the modal window is opened : the connexion is established. As long as the modal window is opened, the connexion is maintained.
The modal window is :



Procedure

The procedure to follow to import materials in a Flux project through the material manager is :

Step	Action	Access
1	From the Flux supervisor, open Flux project	
2	Launch the command : « Import via material manager »	
→	In Flux, a **Mat Manager connexion** window appears to indicate the connexion with the material manager. The material manager is opened. A **Flux database windows** appears with the materials of the Flux project	
3	In the material manager, add the materials in the **Flux database window**	
→	The materials are imported in real time in the Flux project	
4	Close the connexion between Flux and the material manager: • Click on « disconnect » in Flux or • Close the **Flux database windows** 	
→	The connexion is interrupted. The imported materials are available in the Flux project	

8. Few words about Cedrat Distribution Engine

Definition

Cedrat Distribution Engine is a software component enabling distributed computing in Cedrat applications.

Distributed computing consists of distributing computations simultaneously on several processors.
It is carried out on a set of machines, each one of them could be of the multiprocessor type. The use of a single multiprocessor machine is possible.
Distributed computing, compared to sequential computing, allows the reduction of the total computation time when computations can be done concurrently.

Components

Cedrat Distribution Engine consists of two programs:

- A distribution server **Cedrat Distribution Server**: service which allocates computational resources executing and synchronizing the Flux computations on them (scheduler)
- A distribution server management **Cedrat Distribution Manager**: graphical interface of Cedrat Distribution Server management

Functioning

Cedrat Distribution Engine enables the distribution of **Flux** computations run from the optimiser **GOT-It.**

Continued on next page

Compatibility

Here are compatibilities by Flux version:

- Before the Flux version 11.2.0: **Cedrat Distribution Engine 1.0** is compatible with **GOT-It 2.0** and **Flux 11.1 Service Pack 2**.
- The **Flux version 11.2.0** had an update of Java (from the version 1.6 to 1.7). To ensure the compatibility with distribution and optimization tools, a service pack ware necessary (taking into account the update of version Java) for:
  - **Cedrat Distribution Engine 1.0.1**
  - **GOT-It 2.0.1**

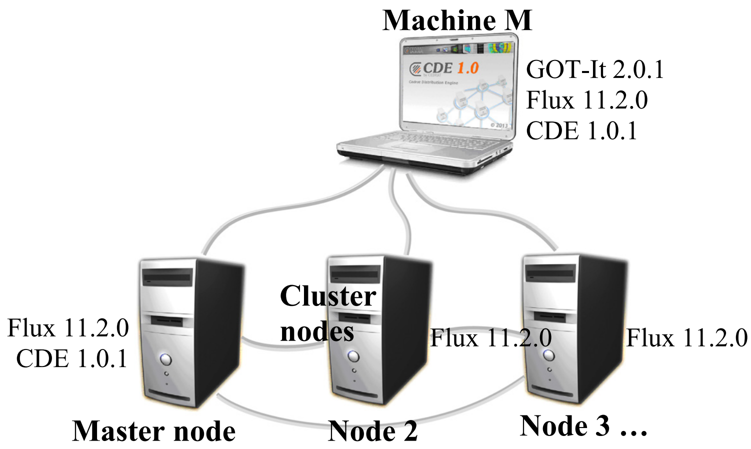
To resume :
The **Flux 11.2.0** is not compatible with **Cedrat Distribution Engine 1.0** and **GOT-It 2.0**
It is necessary to update this both tools with **Cedrat Distribution Engine 1.0.1** and **GOT-It 2.0.1**

Using example with a cluster

An overview of the installation process for a **cluster** is presented in the table below.

Phase	Description	Where
1	GOT-It 2.0.1 installation and license configuration	Machine M
2	Flux 11.2.0 installation and license configuration	
3	Flux 11.2.0 installation and license configuration	Master node*
4	Flux 11.2.0 installation and license configuration	Node 2
5	Flux 11.2.0 installation and license configuration	Node 3 …
6	Cedrat Distribution Engine 1.0.1 installation	Master node*
7	Cedrat Distribution Engine configuration	
8	Cedrat Distribution Engine 1.0.1 installation	Machine M
9	GOT-It configuration to use the distribution	

*Master node:
The cluster node onto which the distribution server and license server will be run is conventionally called the master node.



Notes:

- At the present time, the operating system of each cluster node must be **Windows 64 bits** (for more details, about the supported Windows 64 bits types, you can refer to the Flux installation guide).
- In order to proceed with the software installation, you must log in as **administrator**.

Continued on next page

Using example with single computer

An overview of the installation process on a **single computer** is presented in the table below.

Phase	Description	Where
1	GOT-It 2.0.1 installation	Machine P
2	Flux 11.2.0 instalation and license configuration	
3	Cedrat Distribution Engine 1.0.1 installation	
4	Cedrat Distribution Engine configuration	
5	GOT-It configuration to use the distribution	

Machine P

GOT-It 2.0.1
Flux 11.2.0
CDE 1.0.1

Notes:

- At the present time, the operating system of the machine P must be **Windows 64 bits** (for more details, about the supported Windows 64 bits types, you can refer to the Flux installation guide).
- In order to proceed with the software installations, you must log in as **administrator**.

Warning Memory limitation

Warning:
In the current version, no memory constraint is taken into account. Only the number of processors and license tokens are managed. Consequently, launching multiple Flux instances for which the total memory would exceed the machine running memory limit will lead to a performance drop or impossibility of execution.