Altair

HyperWorks

Altair Flux™ 2019

Release Notes

Introduction

The **Release Note** document gives some useful information on the release of **Altair Flux™ 2019**. It is a technical document in addition to the **New features** document installed with Flux at the following path : *<INSTALLFLUX>\Flux\DocExamples\UserGuide\English\ 01_New_Features_2019.pdf*

Contents

Essential Information

1

Essential information about Flux 2019
For **Flux 2019** the user has the possibility to be protected by "**HWU**" protection (Windows and Linux) **exclusively**. The **Legacy** protection has been **removed**.
In order to use the version HWU, you need: • HW license file Version 19.0 • License Manager-LMX Version 14.0.4
Flux 2019 is available for Windows and Linux OS: • Windows 7, Windows 10 • Red Hat and CentOS 6.6/7.2 • SLES 12 SP2
Flux 2019 takes into account several user feedbacks, and around 40 issues have been **solved** in this version.
Distributed computing with **MPI** is available in Flux for Windows and Linux and can be used with PBS. However, **qualification on Windows cluster** connected with InfiniBand **has not been done and Linux clusters is highly recommended for parallel computing.**
The new feature "**Flux starting guide**" is only available in **Windows** (**not in Linux**).
It is now possible to install Flux on Windows or Linux **without administrator rights** with limited functionalities.
Minimum memory required: • For Flux 2D: 8 GB of RAM • For Flux 3D, Skew and PEEC: 12 GB of RAM
SPEED import is not available since **Flux 2018 version**. Please use a previous Flux version to import your SPEED file.
The **Student Edition of Flux 2019** will be **available** few weeks **after the release of the professional edition**.

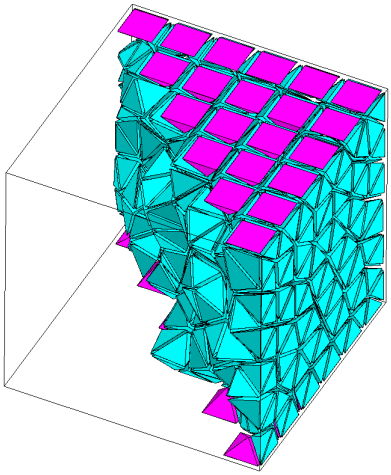
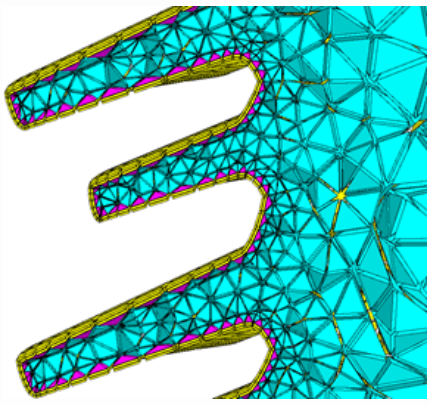
New features of Flux 2019

New features dealing with Environment

New features	Description
Branding	**Flux™** become **Altair Flux™**.
User Mode	A **user mode** has been implemented. Thus the user has the choice between 3 modes: • **Standard**: the standard mode. • **Beta**: gives early access to a set of features which will be officially supported in a future release. • **Advanced**: This mode gives access to a set of expert functionalities and also to beta features (this mode includes the beta mode).
Renaming of the command "Run a python file"	Labels and python command have been changed for running a python file: • **Label** ◦ **Run a python file**: Run a python file with full GUI refresh ◦ **Run a python file in silent mode** : Run a python file with reduced GUI refresh (faster) • **Python command** *Table 1:* See table below **Note:** The "legacy" python commands are always compatible with Flux 2019 and future version.

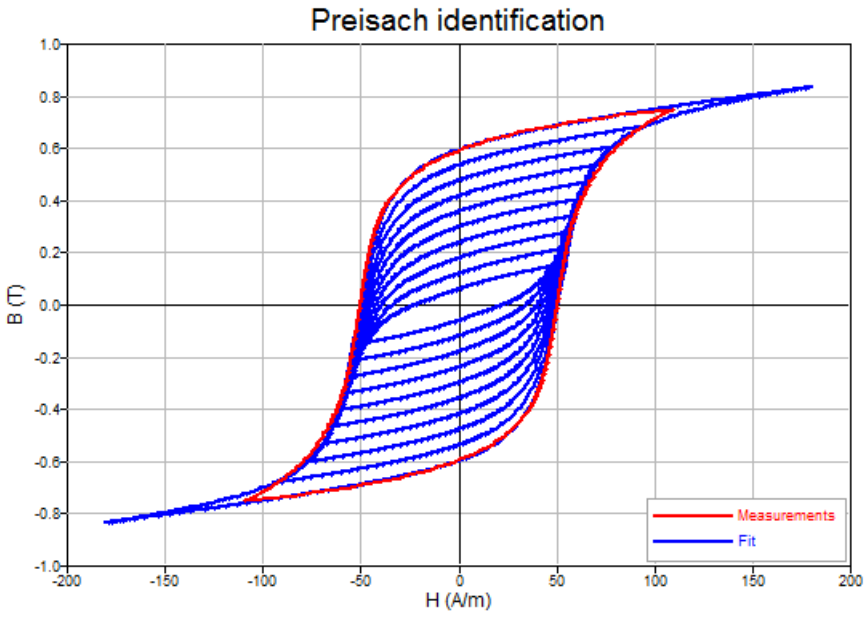
	Before Flux 2019	Starting from Flux 2019
With GUI refresh	executeSpy('file.py')	runPy('file.py')
Without GUI refresh	executeBatchSpy('file.py')	runPyInSilentMode('file.py')

New features dealing with Meshing

New features	Description
Visualize volume elements internal to solids by a cut plane	Possibility to visualize volume mesh element according to a plan defined by 3 points. 
New mesh generator by "Layers"	This mesh generator, based on the MeshGems-Hybrid module of Distene, is a mixed mesh elements generator in Flux (it creates hexahedra, prisms, pyramids, or tetrahedra). It allows to mesh the skin depth with extruded elements (prisms, hexahedra) and relax the mesh in the remaining volume. It is based on the initial surface mesh of the volume without modifying it. 
Mesh Import - wire body	Now, when importing mesh, wire body are also imported into Flux, to allow using line region later.

New features	Description
Improvement of shell elements generation with mesh import	The orientation of the current density on the shell bodies in areas was wrong where the surface elements had been merged during the mesh import. The improvement implemented has been to take into account the correct normal vector on these faces (merged surface elements) in the different Flux algorithms.

New features dealing with Physics

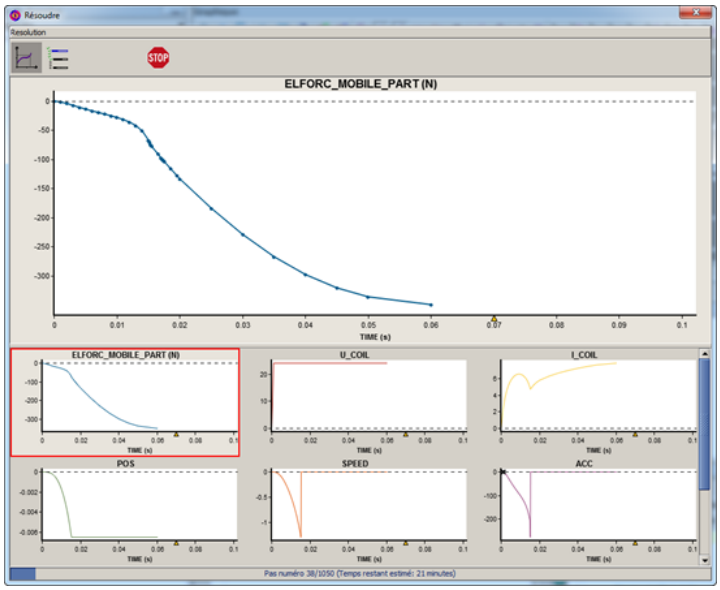
New features	Description
Preisach Model (Hysteresis) *in Beta mode*	A new static vectorial hysteretical model has been developped in Flux in **Beta mode** (Supervisor option). This model is easier to use than the Jiles-Atherton model. The Preisach model is more powerfull to generate minor loops. This model comes in two materials. • A material with four coefficients : Br,Bsat,Hc,s that represent a major loop. • A material with 3*n coefficients intended to fit measurements data (by a fitting tool), in order to be as precise as possible, the fit must be made from the largest cycle performed during the simulation. If we are close to the major loop we identify the major loop, if we are closer to a minor loop it is better to identify this minor loop. 

<table>
<tr><th>New features</th><th>Description</th></tr>
<tr><td></td><td>
Table 2: fitting example: identifiaction of the material M800-50A
<table>
<tr><th>Values</th><th>ai</th><th>bi</th><th>ci</th></tr>
<tr><td rowspan="2">M800-50A</td><td>0.33</td><td>168.2</td><td>174.79</td></tr>
<tr><td>0.65</td><td>27.18</td><td>129.51</td></tr>
</table>
Note: If you want test and use the fitting tool with the new hysterical material, please contact the Support.

To get a better convergence with these materials, some resolution options must be selected :
• Method for computing the relaxation factor for Newton-Raphson:
◦ Maximal factor method (2D and 3D)
• Initialization of state variables at the begining of a time step:
◦ With predicted solution (only 3D)
◦ With the previous step solution (only 2D)
</td></tr>
<tr><td>Speed up of iron losses computation in post processing</td><td>Some improvements have been done with the new iron losses model, especially for the computation time in postprocessing.</td></tr>
</table>

New features dealing with Solving

New features	Description
Result Preview	During the solving, a set of curves are automatically displayed and give the evolution of the main quantities in runtime. If the evolution is wrong, the user can stop directly the solving and adjust his model.

New features	Description
	
Add a Newton-Raphson to fixed point switch for the maximal factor method	For non-linear solving using this relaxation method is now more robust and can improve convergence
Simplification of GUI for solving process options	With Flux 2019, the **Linear system solver** options tab has been simplified. In **Standard** mode, the user has only to choose between 2 types of solver before solving: • **Direct solver** • **Iterative solver** By doing this, the options have been standardized. The older solvers used in the previous version of Flux stay available through the **Advanced** mode. This mode can be set using the supervisor. The filtered options are set to a default value. Solving process options are applicable for all physical applications of Flux 2D, 3D and Skew.
New initialization method by prediction	With Flux 2019, a new initialization method of the state variables at the beginning of each time step is proposed to the user. This method consists in computing a predicted solution of the current time step using previous ones to converge faster.
Matrix size estimator improvement	The estimation of the matrix size could be excessively overestimated, causing a memory-based error message whereas the solving would have been possible. A fix has been done to reduce the cases of excessive overestimation.

New features	Description
Automatic local/ distributed solver strategy	An automatic selection between MUMPS local and distributed is done based on the dimension of the problem and the correctness of the user configuration. User can still use manual selection by using « advanced mode ».
Ensure compatibility with other batch schedulers (parallel computing)	Commands used for parametric distribution through PBS are now accessible and can be modified, making all Altair Flux/PBS features can now be used with other batch schedulers.

New features dealing with Post-processing

New features	Description
3D curve dedicated to rotating machine	The **3D curve dedicated to rotating machine** allows computing a formula values along a computing circle positioned around a rotating machine rotation axis. Those values are collected for a set of time steps (feature available only in **Transient Magnetic** application). Once collected, the values are represented in a 3D curve having in abscises the time and the angular position along the computing circle. A specific option allows eventually to compute those values in 2D Fourier domain depending on the motor-harmonic and the spatial order.

New features about dedicated contexts

New features	Description
Context of Data Import/Export	The **data import/export context** is a Flux environment dedicated to exchanges between Flux and a third party software. It allows making a weak coupling with another application software: • Vibroacoustics • Thermal • Any other physical application This context is implemented in Flux since the 2018 version. It is enriched and more robust at each version. In this 2019 version, a thermal export to OptiStruct has been added.

New features about some added commands

New features	Description
Active scenario	A new command **getActiveScenario()** has been implemented to have the possibility to know which solving scenario is ON when opening Flux with PyFlux. For example, the calling of Flux from SimLab is simplified with this new python command. When opening a solved project that has several scenarios, the post-processing operations point to a given step of a scenario. This new **getActiveScenario()** command allows knowing which one. In a scrip, you have acces to the following command: *result = getActiveScenario()* *print result.name* The instance of the active scenario is returned by the command. Reminder: there is a family of commands to control the post-processing in python (see the document **How to get the steps of a scenario** available from Flux Supervisor)
new APIs about user I/ O parameters (Groovy)	**void stopSolvingASAP()** The user have the possibility to stop a solving process by evaluation of a criterion. Typical example: a sensor reach a wished data. **public boolean isSolving()** The user wants to know if he is in solving or in post-processing. Typical example: in solving, the user compute a value that he wants to store in a file. In post-processing, he wants to read this value in the file. **public int getStepIndex( )** In solving, the user wants to know what is the current step number. Typical example: In solving, l'utilisateur wants to store values in a file, so he must empty the file at the begining of the solving process.

New features about examples

As a reminder, all examples are accessible via the Flux supervisor, in the context of **Open an example**.

New examples	Description
Flux_Compose_Induction_Motor	This example is dedicated to Flux-Compose coupling. The Compose script will perform a linear search to find the input voltage necessary to obtain the nominal current during a rotor blocked test. Once the Flux simulation is done the rest of the process is carried out in Compose.

New features about "How to"

New "How to..."	Description
How to Experiment with NumPy in Flux?	***New*** NumPy is a Python module widely used in scientific computing area and could improve the capacity of what Flux users could achieve when they write PyFlux scripts. However, NumPy is a native C-Python module whereas PyFlux scripts are executed by Jython, a Java Python interpreter, which is not able to execute C Python modules thus preventing NumPy usage with Flux. Nowadays a third-party project named JyNI (Jython Native Interface) provides an experimental solution to execute native C Python with the Jython interpreter, starting with Jython 2.7.1 Since Flux 2018.1 release, Flux has been upgraded to use Jython 2.7.1 and it becomes possible to use the JyNI library to experiment with native C Python modules like NumPy.
How to use Altair Flux™ 2019 with PBS via Display Manager	***Update*** The 2018.0 version of Flux offered the possibility to use PBS Professional Compute Manager to compute distributed jobs. With Altair Flux™ 2019, the possibility to post-process the solved projects located on your cluster is now available through Display Manager. In PBS Display Manager, double-clicking Flux-GUI icon allows you to choose the version and also the application type to open, since the version 2019 of Flux, thanks to the drop menu.
How to launch Altair Flux™ 2019 via command line	***Update*** The three command line options to run python file are renamed: • -runPy replaces -executeSpy • -runPyInSilentMode replaces -executeBatchSpy

New "How to..."	Description
	• -runPyInSilentModeAndExit replaces -executeBatchSpyAndExit
How to set Altair Flux™ 2019 for Batch Schedulers	***New*** This document will provide the necessary knowledge to be able to set Altair Flux up for a canonical use, i.e.: solve one large-scale project using distribution with allocated resources.
How to get the steps of a scenario?	***New*** Some API have been implemented to allow the user to select steps to quickly postprocess. All methods have been added on the structure SCENARIO • isPostprocessingOK() • getValuesParameter() • getIndexStep (String[] parameterNames, double[] values) • selectIndexStep(int index) • selectFirstStep() • selectLastStep() • selectNextStep() • isValidStep(int index)

List of updated macros

Table 3:

Updated Macros	Description
ComputeInductanceMatrix	Compute the inductance matrix of a rotating machine ***Input:*** • A solving scenario with results • The current sources corresponding to the stator supply • The stator coils • The face regions corresponding to the magnets • The face regions corresponding to the magnetic circuit • A name for the resulting project ***Output:*** • Curves corresponding to each inductance

Updated Macros	Description
	• I/O tabulated parameters corresponding to each inductance
SlippingMeanValue	Calculate the slipping Mean value and assigns it to an I/O variation parameter. ***Input:*** • select chosen variation parameter • select the name of the quantity • select number of steps • select solving scenario ***Output:*** • Creates an I/O variation Parameter containing the slipping Mean value
SlippingRMS	Calculate the slipping RMS value and assigns it to an I/O variation parameter. ***Input:*** • select chosen variation parameter • Select the name of the quantity • Select number of steps • select solving scenario ***Output:*** • Creates an I/O variation Parameter containing the slipping RMS value
CreateSensorFor2DMotorSlotForce	Create sensors to compute force on slots for motors ***Input:*** • Select the magnetic region on which you want to compute force (usually rotor, stator or magnets) • Select one starting line (preferably on one extremity) • Select the coordinate system around which the motor is rotating • Indicate “Yes” for radial forces or indicate “No” for tangential forces ***Output:*** • Create cylindrical coordinate system • Create spatial quantity

Updated Macros	Description
	• Create automatically sensors to compute force on slots
CreateSensorFor3DMotorSlotForce	Create sensors to compute force on slots for motor (3D or skew) ***Input:*** • Select the magnetic region on which you want to compute force (usually rotor, stator or magnets) • Select one starting line (preferably on one extremity) • Select the coordinate system around which the motor is rotating • Indicate "Yes" for radial forces or indicate "No" for tangential forces ***Output:*** • Create cylindrical coordinate system • Create spatial quantity • Create automatically sensors to compute force on slots

List of New Macros

Table 4:

New Macros	Description
HalbachMagnetization2D	Generate Halbach magnetization for the selected face regions. A new material "Halbach_magnet" with the desired orientation and remanence induction will be defined and assigned to these face regions. ***Input:*** • Remanent flux density (T) • Magnet relative permeability • Angle for maximum remanent induction (degrees) • Number of pole pairs • Face regions of the magnets with Halbach magnetization ***Output:*** • Create a new material with the desired magnetization profile and assign it to the magnet's face region.

New Macros	Description
HalbachMagnetization3D	Generate Halbach magnetization for the selected volume regions. A new material "Halbach_magnet" with the desired orientation and remanence induction will be defined and assigned to these volume regions. ***Input:*** • Remanent flux density (T) • Magnet relative permeability • Angle for maximum remanence induction (degrees) • Number of pole pairs • Face regions of the magnets with Halbach magnetization ***Output:*** • Create a new material with the desired magnetization profile and assign it to the magnet's volume regions.
Sensor_Display	Display points corresponding to punctual sensors ***Input:*** • Select punctual sensors ***Output:*** • Display points as multi-points support
ExportCurveVsPhaseFromAC	Generate a curve versus phase starting from a complex number defined by modulus and phase ***Input:*** • Formula to export (should be a complex number) • Start angle (°) • End angle (°) • Angle step (°) • File name ***Output:*** • Create a new file
ExportNastranVariousSpeeds	Generate OptiStruct or Nastran files for different speeds for vibro-acoustic analysis ***Input:*** • Angpos rotor parameter

New Macros	Description
	• Case parameter • Number of cases (speeds) • Minimal angle of the scenario • Maximal angle of the scenario • Radius (in m) • Slot opening angle (in degree) • Computation support • Name of force computation • Name of exported files • Overwrite existing force computation • Format (1=Nastran bulk, 2 OS fem, …) • If format =1 or 2, number of harmonics • Export only Global DLOAD ***Output:*** • Create one file of force for each case (speed)
FindOutCornerPoint_Imax_Angle	Find out the corner point of the Torque Vs Speed curve starting from an unresolved Flux Project. ***Input:*** • current Source • value of Imax • value of SpeedMin • value of SpeedMax • the step value of the speed • value of the Vrms max of the inverter • solving Scenario • the user's defined variation parameter for speed • the user's defined variation parameter for the angle • the user's defined variation parameter for the maximal current • value of AngleMin • value of AngleMax • value of Angle Step Value • the extension of the name of the file (for example 'solved') ***Output:*** • variation Parameter containing the corresponding Speed to Vrms max

New Macros	Description
FindOutMaxSpeed_Imax_Angle	find out the Maximal speed in the Torque Vs Speed curve starting from an unresolved Flux Project. ***Input:*** • current Source • value of Imax • value of SpeedMin • value of SpeedMax • the step value of the speed • value of the Vrms max of the inverter • solving Scenario • the user's defined variation parameter for speed • the user's defined variation parameter for the angle • the user's defined variation parameter for the maximal current • value of AngleMin • value of AngleMax • value of Angle Step Value • the extension of the name of the file (for example 'solved') ***Output:*** • Variation Parameter containing the Maximal speed

Resolved Issues of Flux 2019

Resolved Issues

Table 5:

Issues		Short description of the origin problem resolved in this version
Supervisor	FX-8913	Flux supervisor freeze when a cache folder path is entered
Geometry	FX-9710	Problem with complete infinite box. In the two attached projects, the infinite box is created by two ways. In the first project: The infinite box is created by create faces + create volumes. In the second project: The infinite box is created by complete infinite box The 1st project is solved with a warning concerning electric cut loop (and it is true, it is not necessary to create it), the 2nd one can not be solved because of this error message. Just to remind that the two projects are the same, It is a body car surrounded by air.
Mesh	FX-12151	Two identical magneto-static project gives different results The two projects are identical, but the results obtained in the two projects are differents.
Mesh	FX-12010	Bad visualization of mapped mesh on a cut plane when we want to visualize mesh on cut plane , and a volume is build only with parralepiped mesh element (mapped and extrusive), the mesh on the cut plane is done with tetra only.
Mesh	FX-7909 FX-8142	have a possibility to better display mesh inside 3D volume
Mesh	FX-12160	In the case of 3D overlay (SPM motor and IM motor), we activate advancing front (Netgen mesher) mesh generator. Once the project is created, it is impossible to modify the mesh option in order to set MeshGems option.
Circuit	FX-11957	In this project with mesh import and with terminal of solid conductor assigned to faces, the electric circuit is delete if we enter in the circuit editor context.

Issues		Short description of the origin problem resolved in this version
Circuit	FX-10712	Interface problem in the menu circuit : when a circuit is deleted, the menu to import circuit doesn't appears anymore.
Circuit	FX-8521	In Flux if we have defined a sensor of Flux associated to a stranded coil conductor component of the circuit, the association is broken if we enter in the circuit editor. In a SENSOR_1 is associated to the stranded coil conductor COIL_CONDUCTOR_1, if we open and close the circuit editor context SENSOR have no more stranded coil conductor
Material	FX-11397	Curve for Materials with Spline uses non linear interpolation while during solving the real curve that is used is a linear interpolation between points. This is very confusing for customers : they see "Spline" and they can plot it as a nice smooth curve whereas during solving it will be considered as a list of segments.
Memory	FX-10516	MUMPs memory limitation without warning When MUMPs RAM memory is limited by the user (see solving process options in the attached image), an error arises if the required RAM memory is over the limitations. The same error occurs when the computer has reached its own RAM limit. Since both situations are different (one can be directly solved by the user within Flux and the other not), I think it will be convenient display a warning message when this limitation may be too low for the expected MUMP requirements.
Solving	FX-7769	When we stop the simulation and when we do continue solving the process, Flux display an error message
Solving	FX-9696	Impossible to solve the projet in presence of the path
Distribution	FX-11399 FX-11347 FX-11276	Impossible to distribute with one geom parameter and the angular position
PyFlux command	FX-11429	Have the possibility to know which solving scenario is ON when opening Flux with PyFlux. It will simplify the calling of Flux from SimLab.

Issues		Short description of the origin problem resolved in this version
PyFlux command	FX-8535	Add compute method for evolutionFormula When EvolutioFormula contains a PARAMETER (example EvolutionFormula[10].FORMULA = '360/PARAMETRE'), it's impossible to evaluate the formula before solving. In Python, we have often this issue.
Coupling	FX-11673	Flux-Flux coupling (Magneto-Thermal coupling) The magnetic project of this Flux-Flux coupling project (magneto-thermal simulation) presents an error at the end of the resolution but only if the multiphysics solving session is activated. Otherwise, the same scenario is solved correctly.
Coupling	FX-11882	Import/Export context takes a long time to import a mesh in solved projects Using the tutorial Vibro to OS (WoundRotor case) the import of mesh takes 4 minutes while it is instantaneous in the mechanical analysis context. Good news is that it only takes 1 sec if you do it before solving. So it must do checks on all solving steps. Please also check the deletion process.
Macro	FX-11647	Wrong behavior of the ComputeInductanceMatrix macro The python generated by the ComputeInductanceMatrix macro and by extension the computation are corrupted. The problem is due to the python interpretor which replaces the list of the instances by the key word ALL on the fields CURRENTSUPPLY and COILS
Postprocessing	FX-11182	Update the video compiler for animations
Documentation	FX-11460	Add documentation on "Distribution Manager" and "Server Manager" boutons from the supervisor Add information in the installation guide
Documentation	FX-7630	give user facility to open memento (from supervisor for instance)
Documentation	FX-8540	Add a better link to the list of macro
Label	FX-11557	French labels "Plan de reference" in the modeler

About external software – supported version

4

External programs

The use of external programs can be achieved with the versions listed in the table below.

Table 6:

Flux functionalities	External software	Supported version	Supported OS	Flux OS Win64	Flux OS Lin64
Flux - Simulink Coupling	Matlab - Simulink	2016b	64 bits	✓ Flux 12.2 Flux 12.3 Flux 2018 Flux 2018.1 Flux 2019	✗
		2017a & 2017b		✓ Flux 12.3 Flux 2018 Flux 2018.1 Flux 2019	✗
		2018a & 2018b		✓ Flux 2019	✗
Distribution computing	CDE	2.2	32 bits (can be installed in a 64 bits to exchange with Flux)	✓	✗

External Altair programs

External programs coupled with Flux 2019.

Table 7:

Functionalities	Altair software	Compatible version with Flux 2019
Export to .. (Vibroacoustics and mechanical analysis)	OptiStruct	2019 version and more
Export to.. (Thermal analysis)	AcuSolve	
Co-simulation	Activate	
Mesh import	HyperMesh	
Mesh import	SimLab	
Optimization	HyperStudy	

About protection and installation

5

Since this 2019 version, there only one Flux installer, the HWU licensing system. The Legacy licensing system is no longer released.

About HWU

Since the 12.2 version, it is possible to use the HyperWorks Units as protection system. The HWUs count is split into 2 parts:

- GUI = description of geometry+ mesh + physics + postprocessing of results
- Solver = computation of the model

All the applications are covered.

HWU for Flux 3D/ Skew / PEEC

For Flux 3D/Skew/PEEC, the HWU licensing system is the same as all HyperWorks solvers:

- GUI = 21 HWUs
- Solver = 30 HWUs

HWU for Flux 2D

For 2D users exclusively, the offer has been adapted. There is no GUI / Solver distinction. By default, the user has access to all applications and with 15 HWUs.

Documentation installation

The setup specific to the documentation is integrated on the main setup of Flux. Since Flux 11.2, documents and examples are automatically installed with the main setup of Flux.

About installation - CDE and CSS

The Computing Distribution Engine tool (CDE) and the Computing Soft Server (CSS) allowing distributing computations, are not installed with the main setup of Flux. The user shoud install its starting from the supervisor (see the installation guide).

Graphics cards

It is necessary for the proper functioning of our software that the driver of the graphics card is as up-to-date as possible. In our experience a "Windows Update" is not sufficient, it is essential to install the latest driver supplied by the manufacturer of the card.

Eg for NVIDIA: http://www.nvidia.com/Download/index.aspx?lang=en-us