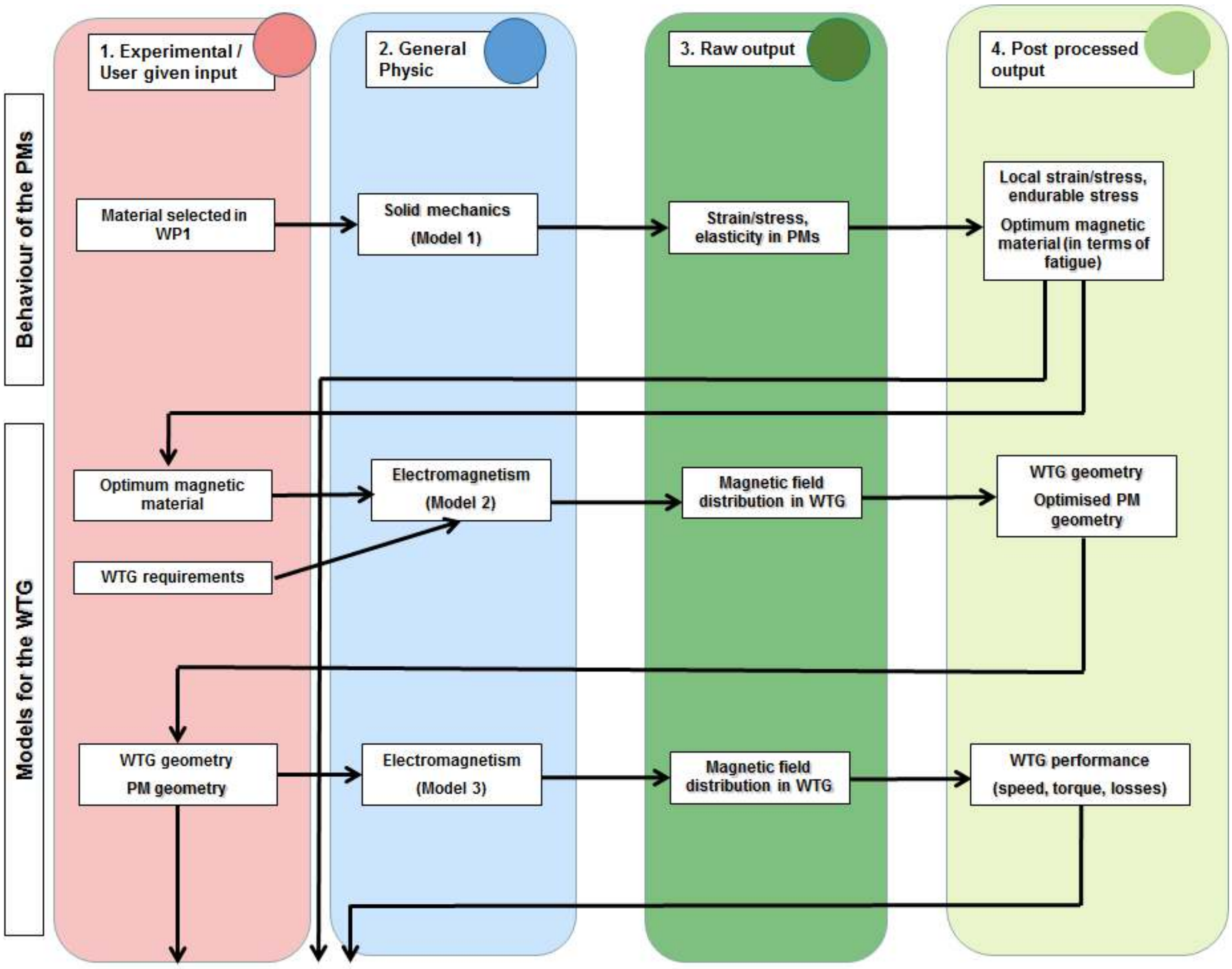
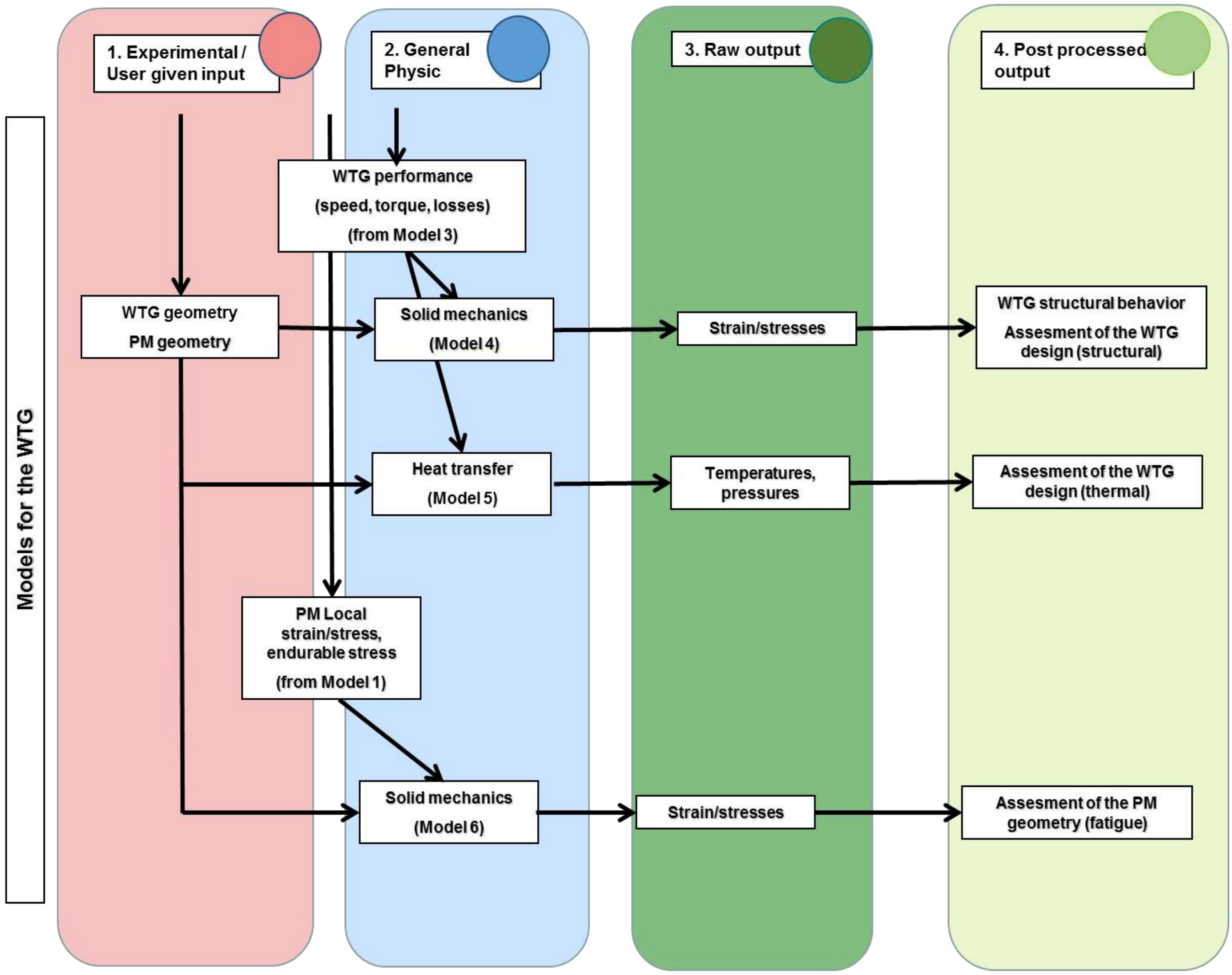
NEOHIRE: NEOdymium-Iron-Boron base materials, fabrication techniques and recycling solutions to HIghly REduce the consumption of Rare Earths in Permanent Magnets for Wind Energy Application

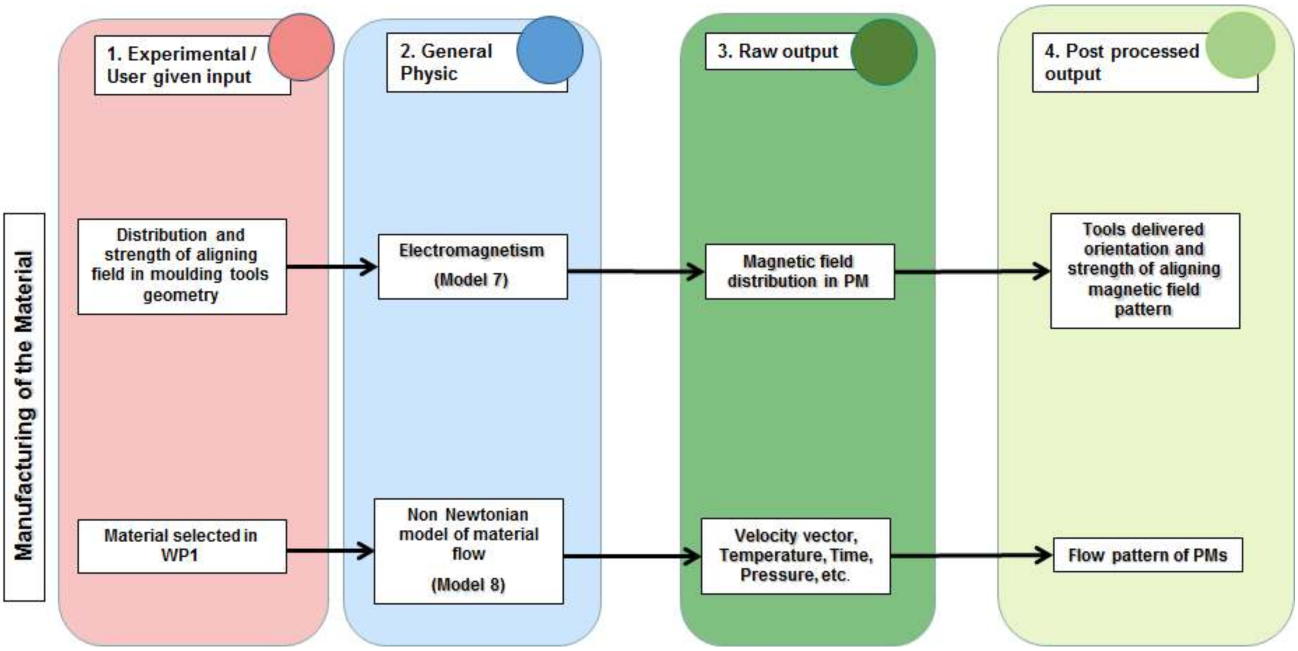
OVERVIEW of the simulation			
1	USER CASE	NEOHIRE main objective is to reduce the use rare earth elements in the permanent magnets (PM) present in wind turbine generators (WTG). For this purpose, new PM geometries and WTG designs, as well as new anisotropic aligning tools (AT) for processing and flow of the thermoplast based anisotropic NdFeB HDDR powder composites (MF) will be modelled and simulated.	
2	CHAIN OF MODELS	MODEL 1	Continuum model - Solid mechanics applied to stress distribution of permanent magnet fatigue specimens
		MODEL 2	Continuum model - Electromagnetic model applied to WTG active parts for electromagnetic predesign
		MODEL 3	Continuum model - Electromagnetic model for detailed simulation of WTG
		MODEL 4	Continuum model - Structural mechanics model of the WTG
		MODEL 5	Continuum model - Heat flow model for the analysis of the WTG
		MODEL 6	Continuum model - Solid mechanics applied to stress distribution of PM in WTG
		MODEL 7	Continuum model (Electromagnetism, magnetics) - Model for the design of the active parts of the moulding tools for aligning the magnetic material
		MODEL 8	Continuum model (Continuum mechanics, Fluid mechanics) Non Newtonian model of material flow
3	PUBLICATION ON THIS ONE SIMULATION	N/A	
4	ACCESS CONDITIONS	All models will be treated as "confidential" The software-tools for the simulations are all commercial	

WORKFLOW



1. Experimental / User given input
2. General Physic
3. Raw output
4. Post processed output
Behaviour of the PMs
Models for the WTG
Material selected in WP1
Solid mechanics (Model 1)
Strain/stress, elasticity in PMs
Local strain/stress, endurable stress
Optimum magnetic material (in terms of fatigue)
Optimum magnetic material
WTG requirements
Electromagnetism (Model 2)
Magnetic field distribution in WTG
WTG geometry
Optimised PM geometry
WTG geometry
PM geometry
Electromagnetism (Model 3)
Magnetic field distribution in WTG
WTG performance
(speed, torque, losses)


1. Experimental / User given input
2. General Physic
3. Raw output
4. Post processed output
Models for the WTG
WTG performance (speed, torque, losses) (from Model 3)
WTG geometry
PM geometry
Solid mechanics (Model 4)
Strain/stresses
WTG structural behavior
Assesment of the WTG design (structural)
Heat transfer (Model 5)
Temperatures, pressures
Assesment of the WTG design (thermal)
PM Local strain/stress, endurable stress (from Model 1)
Solid mechanics (Model 6)
Strain/stresses
Assesment of the PM geometry (fatigue)


1. Experimental / User given input
2. General Physic
3. Raw output
4. Post processed output
Manufacturing of the Material
Distribution and strength of aligning field in moulding tools geometry
Electromagnetism (Model 7)
Magnetic field distribution in PM
Tools delivered orientation and strength of aligning magnetic field pattern
Material selected in WP1
Non Newtonian model of material flow (Model 8)
Velocity vector, Temperature, Time, Pressure, etc.
Flow pattern of PMs

MODEL 1: Solid mechanics applied to stress distribution of permanent magnet fatigue specimens

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED	
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	Analysis of the strains/stresses, stress gradients, highly stressed volume of partic specimens in order to assess the correlation between local existing stresses and endurable stresses of the experimental investigation by means of linear elastic calculations (for fatigue assessment)
1.2	MATERIAL	Magnetic Materials to be selected in WP1
1.3	GEOMETRY	Unnotched and notched small scale specimens to be defined in WP2
1.4	TIME LAPSE	To be defined in the project
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	The highly stressed areas of specimen manufactured in different material states respectively manufacturing routes will be analysed under axial and probably bending loading according to procedure of the experimental material characterization (fatigue).
1.6	PUBLICATION ON THIS ONE SIMULATION	N/A

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Solid mechanics: Static strain/stress analysis, linear-elastic calculation	
2.1	MODEL ENTITY	Finite-Element-	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	Equation	F = k · s
		Physical quantities	displacement, stiffness, force
2.3	MATERIALS RELATIONS	Relation	Linear elasticity: [σ] = [E] · [ε]
		Physical quantities/ descriptors for each MR	strain/stress, elasticity
2.4	SIMULATED INPUT	"N/A (input data obtained via experiments in WP2)".	

3	SOLVER AND COMPUTATIONAL TRANSLATION OF THE SPECIFICATIONS		
3.1	NUMERICAL SOLVER	FEM, linear-elastic calculations	
3.2	SOFTWARE TOOL	Abaqus	
3.3	TIME STEP	To be defined in the project	
3.4	COMPUTATIONAL REPRESENTATION ,	PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL	[σ] = [E] · [ε] Young's modulus, Poisson's ratio
3.5	COMPUTATIONAL BOUNDARY CONDITIONS	According to load cases expected during WTG operation (to be defined in the project)	
3.6	ADDITIONAL SOLVER PARAMETERS	--	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	**POST PROCESSING**	
4.1	**THE PROCESSED OUTPUT**	Correlation between local strain/stress and endurable stress to be used in **Model 6** Selection of optimum magnetic material (at least for fatigue) for the PM design in the WTG in **Models 2-8**.
4.2	**METHODOLOGIES**	Splining through raw output data
4.3	**MARGIN OF ERROR**	Depending on scatter of material and fatigue investigated

MODEL 2: Electromagnetic model applied to WTG active parts for electromagnetic predesign

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	The user wants to define different PM geometries, together with the active parts (stator and rotor) of a WTG, in order to optimize the performance of such a generator in terms of power per unit weight of CRM.
1.2	MATERIAL	PM materials to be selected in WP2. Industrial-grade non-oriented electrical steel. Industrial-grade copper conductors.
1.3	GEOMETRY	Geometry to be defined in WP3.
1.4	TIME LAPSE	To be defined in the project.
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	Requirements of WTG specified in Task 3.1 of WP3.
1.6	PUBLICATION ON THIS ONE SIMULATION	N/A

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Continuum electromagnetic model	
2.1	MODEL ENTITY	Finite volumes	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	**Equation**	Maxwell's Equations simplified for low frequency fields (displacement current term ignored) with no consideration of the electric polarization phenomenon (Gauss's law ignored): $\nabla \times \mathbf{E} = -\partial_t \mathbf{B}$ $\nabla \cdot \mathbf{B} = 0$ $\nabla \times \mathbf{H} = \mathbf{J}$
		Physical quantities	Electric field (E). Current density (J). Magnetic flux density (B). Magnetic field intensity (H).
2.3	MATERIALS RELATIONS	**Relation**	<u>Magnetic relations</u>: biunivocal B-H relation for PMs (linear, isotropic), copper (linear, isotropic) and electrical steel (spline curve fitted from manufacturer data, anisotropic). Minor hysteresis loops ignored for PMs. Hysteresis loops and induced eddy-currents ignored in the field solution for the electrical steel (added later as a post-processing result by considering power loss relationships; see below). <u>Electrical relations</u>: Ohm's law for PMs and copper: $\mathbf{J} = \sigma \mathbf{E}$ <u>Thermal relations</u>: PM B-H relation dependent on magnet temperature (magnet remanence and intrinsic magnetic field coercitivity dependent on temperature, isotropic) (obtained via experiments in WP2). PM and copper electrical conductivities dependent on temperature (obtained via experiments and manufacturer data in WP2, isotropic).

2	Generic Physics Of The Model Equation		
		Physical quantities/ descriptors for each MR	Electric field (E). Current density (J). Magnetic flux density (B). Magnetic field intensity (H). Electrical conductivity (σ). Frequency of the main harmonic of the flux density waveform (f) Electrical steel magnetic loss density (kW/kg from manufacturer data).
2.4	Simulated input	Optimum magnetic material (at least for fatigue) postprocessed after **Model 1.**	

3	Solver and Computational translation of the specifications		
3.1	Numerical Solver	PE (Simplified Maxwell's Equations) (see 2.2) further simplified in order to describe the magnetic field solution in terms of lumped parameters for the different WTG parts (rotor and stator). Use of the concepts of Magnetomotive Force, Magnetic Reluctance and Magnetic Flux (electrical circuit analogy). Direct algebraic solution of the magnetic field by considering such a lumped parameter approach. Variable separation method applied to the underlying Poisson Problem (Simplified Maxwell's Equations) to certain WTG parts (stator slots and air-gap). Field solution given in terms of sum of harmonic functions.	
3.2	Software tool	Code developed by CEIT in MathWorks MATLAB language.	
3.3	Time step	According to WTG design requirements.	
3.4	Computational Representation	Physics Equation, Material Relations, Material	Physical quantities dependent on WTG geometry local coordinates (i.e. Maxwell's Equations are solved for the whole WTG active parts, but for each part (e.g. PM, electrical steel, copper, air), different properties are considered.
3.5	Computational boundary conditions	Electromagnetic boundary conditions (isolating, continuity, etc).	
3.6	Additional Solver Parameters	N/A	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	POST PROCESSING	
4.1	THE PROCESSED OUTPUT	Raw output: Electromagnetic field solution (a.k.a. magnetic flux distribution): knowledge of the local values of the following physical quantities: electric field (E), current density (J), magnetic flux density (B), magnetic field intensity (H). Electromagnetic torque and output voltages and currents generated by the WTG. Power losses of the WTG. WTG efficiency. WTG power per unit weight of CRM. Optimised PM and active parts to be used in **Models 3-8.**
4.2	METHODOLOGIES	Computed from classical electrical and magnetic circuit concepts (flux linkage, electrical voltage drop, electromagnetic torque, etc.). Hysteresis and induced eddy-current power losses in the electrical steel laminations dependent on magnetic flux density waveform (Generalized Bertotti loss model from manufacturer data): $P_{loss} = k_h B_{max}^2 f + k_e \left\langle (\partial_t B)^2 \right\rangle + k_a \left\langle (\partial_t B)^{3/2} \right\rangle$
4.3	MARGIN OF ERROR	To be defined in the project

MODEL 3: Electromagnetic model for detailed simulation of WTG

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	The user wants to analyse the electromagnetic behaviour of the different PM shapes selected in WP2 , when they are working inside of a WTG
1.2	MATERIAL	Materials selected in WP1
1.3	GEOMETRY	Geometry selected in Task 3.1 of WP3
1.4	TIME LAPSE	To be defined in the project.
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	Requirements of WTG specified in Task 3.1 of WP3
1.6	PUBLICATION ON THIS ONE SIMULATION	N/A

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Continuum electromagnetic model	
2.1	MODEL ENTITY	Finite elements	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	Equation	Maxwell's Equations $\nabla \cdot \mathbf{D} = \rho \quad \nabla \times \mathbf{E} = -\partial_t \mathbf{B}$ $\nabla \cdot \mathbf{B} = 0 \quad \nabla \times \mathbf{H} = \mathbf{J} + \partial_t \mathbf{D}$
		Physical quantities	Electric field (E) Electric displacement field (D) Current density (J) Magnetic flux density (B) Magnetic field intensity (H)
2.3	MATERIALS RELATIONS	Relation	There are no further materials relations requiring additional equations The electrical relation is described by: $\mathbf{D} = \varepsilon \mathbf{E}$ The magnetic relation is described by: $B = \mu H$ Thermal relations: 1) there is a temperature dependence of nonlinear B-H curves for the permanent magnets, and 2) the electrical conductivity of the materials is depending on temperature; these relationships are obtained experimentally in WP2.
		Physical quantities/ descriptors for each MR	Electrical conductivity: (σ) Permeability (μ) Electric field (E) Electric displacement field (D) Magnetic flux density (B) Magnetic field intensity (H)
2.4	SIMULATED INPUT	N/A	

3	SOLVER AND COMPUTATIONAL TRANSLATION OF THE SPECIFICATIONS		
3.1	NUMERICAL SOLVER	PE	
3.2	SOFTWARE TOOL	COMSOL	
3.3	TIME STEP	According the requirements	
3.4	COMPUTATIONAL REPRESENTATION	PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL	Written up for finite elements
3.5	COMPUTATIONAL BOUNDARY CONDITIONS	Electromagnetic boundary conditions (isolating, continuity, etc)	
3.6	ADDITIONAL SOLVER PARAMETERS	Pure internal numerical solver details, If applicable, like • Convergence control algorithm • Time incrementation scheme	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	POST PROCESSING	
4.1	THE PROCESSED OUTPUT	Electromagnetic assessment of the PM selected in WP3 working inside the WTG.
4.2	METHODOLOGIES	Computed from classical electrical and magnetic circuit concepts.
4.3	MARGIN OF ERROR	To be defined in the project

MODEL 4: Structural mechanics model of WTG critical components

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED	
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	The user wants to analyse the status of the stress and strain field in the critical components (such as Axle or bearings) taking into account the geometrical modifi and the most critical load cases. Therefore, a frequency response analysis will be carried out in order to know the natural frequency of system and allow us to avoid resonance problems.
1.2	MATERIAL	Materials selected in WP1
1.3	GEOMETRY	Geometry selected in Task 3.1 of WP3
1.4	TIME LAPSE	Studying the information collected through one year of WTG use (regarding loads, momentums), the load cases analyzed will be the most frequent case (normal use) and some overload cases.
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	In case of frequency study: imposed frequencies In case of stress-strain study, external forces
1.6	PUBLICATION ON THIS ONE SIMULATION	N/A

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Solid mechanics: Static stress analysis procedures and Natural frequency extraction	
2.1	MODEL ENTITY	Finite element	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	**Equation**	The static stress analysis applied, will find an approximate (finite element) solution for the displacements, deformations, stresses, forces. The exact solution of such problem requires that both force and moment be maintained in equilibrium over a finite number of divisions of the volume of the body. The exact equilibrium statement is written in the form of virtual work statement. Generally, We will use Newton´s method as a numerical technique for solving the nonlinear equilibrium equations, although modified Newton or quasi-Newton methods can be used. The model of plasticity used (incremental plasticity theory) is based on a basic fundamental postulate. The inelastic response models the elastic and inelastic responses are distinguished by separation the deformation into recoverable (elastic) and nonrecoverable (inelastic) parts. A more general assumption the total deformation F $F=F^{el}.F^{pl}$ is made up of inelastic deformation followed by purely deformation
		Physical quantities	Time, displacement, stress, strain, plastic strain, natural frequency, accelerations, velocity
2.3	MATERIALS RELATIONS	**Relation**	Force, Energies, Frequencies, Stress-strain field
		Physical quantities/ descriptors for each MR	Force fields in the components Energies field Frequencies field in the system Stress-strain field

2.4	SIMULATED INPUT	Input parameters regarding the maximum torque/load in normal working and overload loadcase, the frequency of the stator-rotor subsystem.

3	SOLVER AND COMPUTATIONAL TRANSLATION OF THE SPECIFICATIONS		
3.1	NUMERICAL SOLVER	The finite element models are usually nonlinear and can involve from a few to thousands of variables. In terms of these variables the equilibrium equations obtained by discretizing the virtual work equation	
3.2	SOFTWARE TOOL	Abaqus 6.14	
3.3	TIME STEP	Depends on the type of analysis. It will be defined in the project	
3.4	COMPUTATIONAL REPRESENTATION	PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL	FN(uM)=0 where FN is the force component conjugate to the NTH variable in the problem and uM is the value of the MTH variable. The basic problem is to solve the equation above for the uM throughout the history of interest
3.5	COMPUTATIONAL BOUNDARY CONDITIONS	It will depend on the load case of study	
3.6	ADDITIONAL SOLVER PARAMETERS	Pure internal numerical solver details, If applicable, like • Convergence control algorithm • Time incrementation scheme	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	POST PROCESSING	
4.1	THE PROCESSED OUTPUT	The raw outputs are: 1. All physical displacement components, including rotations at nodes with rotational degrees of freedom 2. Field of stress, strain, plastic strain velocities and frequencies of the nodes/integration points within the finite element, which is part of the component 3. This is post processed into values for the rest of the components. Based on this a selection of the optimal component geometry is chosen
4.2	METHODOLOGIES	The results fields are calculated in the integration points or nodes and then translated to the rest of the component selection criteria
4.3	MARGIN OF ERROR	Typically less than 5%

MODEL 5: Heat flow model applied to WTG

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED	
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	Heat transfer problems involving conduction, forced convection, and boundary radiation can be analyzed. In these analyses the temperature field is calculated and the stress/deformation state is evaluated in the bodies being studied taking into account the temperature change
1.2	MATERIAL	Materials selected in WP1
1.3	GEOMETRY	Geometry selected in Task 3.1 of WP3
1.4	TIME LAPSE	Studying the information collected through one year of WTG use (regarding temperatures, loads and momentums), the load cases analyzed will be the most frequent case (normal use) and some overload cases.
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	Input parameters regarding the field of temperatures that implies thermal dilatations in normal working and overload loadcases.
1.6	PUBLICATION ON THIS ONE SIMULATION	N/A

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Heat transfer equation	
2.1	MODEL ENTITY	Finite element	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	**Equation**	$-\left(\frac{\partial qx}{\partial x}+\frac{\partial qy}{\partial y}+\frac{\partial qz}{\partial z}\right)+Q=\rho c\frac{\partial T}{\partial t}$
		Physical quantities	Integration point temperatures, magnitude and components of the heat flux vector, current values of uniform distributed heat fluxes, Nodal point temperatures
2.3	MATERIALS RELATIONS	**Relation**	Current values of uniform distributed heat diffusivity Magnitude of the heat diffusivity which PE it completes Temperature degree of freedom n at a node
		Physical quantities/ descriptors for each MR	Temperature field Heat fluxes field in the system
2.4	SIMULATED INPUT		

3	SOLVER AND COMPUTATIONAL TRANSLATION OF THE SPECIFICATIONS		
3.1	NUMERICAL SOLVER	The finite element models are usually nonlinear and can involve from a few to thousands of variables. In terms of these variables the equilibrium equations obtained by discretizing the virtual work equation	
3.2	SOFTWARE TOOL	Abaqus 6.14, LS dyna	
3.3	TIME STEP	If applicable	
3.4	COMPUTATIONAL REPRESENTATION	PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL	The heat transfer rate is expressed by: Q=-kA(ΔT/Δx) The convection/diffusion process uses the trapezoidal rule for time integration. They include numerical diffusion control (the "upwinding" Petrov-Galerkin method) and, optionally, numerical dispersion control
3.5	COMPUTATIONAL BOUNDARY CONDITIONS	Real boundary conditions, trying to be as close as possible to the reality. Boundary conditions are very often nonlinear, film coefficients can be functions of surface temperature, the nonlinearities are often mild and cause little difficulty	
3.6	ADDITIONAL SOLVER PARAMETERS	Pure internal numerical solver details, If applicable, like • Convergence control algorithm • Time incrementation scheme • The software automatically determines a suitable increment size for each increment of the step. • Regarding the number of DOF we would suggest an initial "time" increment and define a "time" period for the step	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	POST PROCESSING	
4.1	THE PROCESSED OUTPUT	Nodal point temperatures in the whole WTG (check for temperature raise in WTG critical components, e.g: PMs, windings, bearings, etc.).
4.2	METHODOLOGIES	The results fields are calculated in the integration points or nodes and then translated to the rest of the component
4.3	MARGIN OF ERROR	Typically less than 2%

MODEL 6: Solid mechanics applied to stress distribution of PM in WTG

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	Analysis of the strains/stresses, stress gradients, highly stressed volume of the design of the PM considered for the WTG by means of linear elastic calculations and fatigue assessment with regard to lifetime respectively allowable stresses based on the approach of Model 1
1.2	MATERIAL	Finally selected magnetic material
1.3	GEOMETRY	Proposed PM geometry to be used in the WTG
1.4	TIME LAPSE	time independent
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	Assessment of the stress/strain distribution and the highly stressed areas of prototypes under service load conditions in order to guarantee a reliable service with regard to fatigue
1.6	PUBLICATION ON THIS ONE SIMULATION	

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Static strain/stress analysis, linear-elastic calculation	
2.1	MODEL ENTITY	Finite-Element-Modelling	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	Equation	F = k · s
		Physical quantities	displacement, stiffness, force
2.3	MATERIALS RELATIONS	Relation	Linear elasticity: [σ] = [E] · [ε]
		Physical quantities/ descriptors for each MR	strain/stress, elasticity
2.4	SIMULATED INPUT	Correlation between local strain/stress and endurable stress, output of Model 2, Calculated structural behaviour of the WTG especially of the PM component, Model 4	

3	SOLVER AND COMPUTATIONAL TRANSLATION OF THE SPECIFICATIONS		
3.1	NUMERICAL SOLVER	FEM, linear-elastic calculations	
3.2	SOFTWARE TOOL	Abaqus	
3.3	TIME STEP	time independent	
3.4	COMPUTATIONAL REPRESENTATION	PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL	[σ] = [E] · [ε] Young's modulus, Poisson's ratio
3.5	COMPUTATIONAL BOUNDARY CONDITIONS	PM and WTG geometry and experimental test set-up	
3.6	ADDITIONAL SOLVER PARAMETERS	--	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	POST PROCESSING	
4.1	THE PROCESSED OUTPUT	Fatigue assessment respectively lifetime estimation
4.2	METHODOLOGIES	Local fatigue concept
4.3	MARGIN OF ERROR	Depending on input calculations, scatter of material and fatigue properties Estimation of the deviation of the numerical assessment by experimental validation of the PM prototype by means of experimental investigation (fatigue)

MODEL 7: Model for the design of the active parts of the moulding tools for aligning the magnetic material

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED	
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	Analysis of the magnetic field strength and distribution in anisotropic moulding tools for material processed in WP1
1.2	MATERIAL	Materials selected from hard, soft and non magnetic database with special abrasion resistivity
1.3	GEOMETRY	Test specimens defined in WP2 and test specimens defined in WP3
1.4	TIME LAPSE	At start of WP2 (for test specimens) and in parallel with the experimental material characterisation (for WGT defined geometry). None (Field is geometry and material characteristics dependent)
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	Depending on the aligning field required for material processed in WP1 and WP2.
1.6	PUBLICATION ON THIS ONE SIMULATION	NA

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Continuum model – Electromagnetism (magnetics)	
2.1	MODEL ENTITY	Finite-Element-Modelling	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	**Equation**	Maxwell's equations: $\nabla \cdot \mathbf{D} = \rho \quad \nabla \times \mathbf{E} = -\partial_t \mathbf{B}$ $\nabla \cdot \mathbf{B} = 0 \quad \nabla \times \mathbf{H} = \mathbf{J} + \partial_t \mathbf{D}$
		Physical quantities	Magnetic flux density (B) Magnetic field intensity (H) Magnetic permeability (?)
2.3	MATERIALS RELATIONS	**Relation**	There are no further materials relations requiring additional equations
		Physical quantities/ descriptors for each MR	Relative permeability Magnetic flux density Magnetisation saturation
2.4	SIMULATED INPUT	Distribution and strength of aligning field in moulding tools geometry	

3	SOLVER AND COMPUTATIONAL TRANSLATION OF THE SPECIFICATIONS		
3.1	NUMERICAL SOLVER	FEM, magnetics	
3.2	SOFTWARE TOOL	Maxwell 14	
3.3	TIME STEP	Mesh dependent. (Standard iteration up to an hour)	
3.4	COMPUTATIONAL REPRESENTATION	PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL	Derived from Maxwell equations (inbuilt for magnetostatics and transient))
3.5	COMPUTATIONAL BOUNDARY CONDITIONS	Depends on the boundary conditions of experimental test set-up (geometry, field components).	
3.6	ADDITIONAL SOLVER PARAMETERS	Optimetrical	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	POST PROCESSING	
4.1	THE PROCESSED OUTPUT	Tools delivered orientation and strength of aligning magnetic field pattern
4.2	METHODOLOGIES	Processing with resolver viewer
4.3	MARGIN OF ERROR	Mesh dependent (default 1%)

MODEL 8: Non Newtonian model of material flow

1	ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED	
1.1	ASPECT OF THE USER CASE TO BE SIMULATED	The user wants to analyze the material flow and solidification in processing of compound from WP2 for prototypes in WP3
1.2	MATERIAL	Material processed in WP2 (thermoplast/HDDR NdFeB composite compound)
1.3	GEOMETRY	Geometry selected in WP3
1.4	TIME LAPSE	Before prototype moulding (not on project critical path) From seconds to minutes (time to solidify material in magnetic field)
1.5	MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS	Requirements of WTG magnet geometry from WP3
1.6	PUBLICATION ON THIS ONE SIMULATION	N/A

2	GENERIC PHYSICS OF THE MODEL EQUATION		
2.0	MODEL TYPE AND NAME	Continuum model (Continuum mechanics, Fluid mechanics) Non Newtonian model of material flow	
2.1	MODEL ENTITY	Finite elements	
2.2	MODEL PHYSICS/ CHEMISTRY EQUATION PE	**Equation**	3D flow motion: $\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \mathbf{u} = 0$ $\frac{\partial}{\partial t}(\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u u} - \boldsymbol{\sigma}) = \rho \mathbf{g}$ $\boldsymbol{\sigma} = -p\mathbf{I} + \eta(\nabla \mathbf{u} + \nabla \mathbf{u}^T)$
		Physical quantities	Velocity vector, Temperature, Time, Pressure, Total stress tensor, Density Viscosity, Thermal conductivity, Specific heat, Shear rate
2.3	MATERIALS RELATIONS	**Relation**	There are no further materials relations requiring additional equations
		Physical quantities/ descriptors for each MR	For viscosity model (modified Cross Model): $\eta = \frac{\eta_0}{1 + \left(\frac{\eta_0 \dot{\gamma}}{\tau^*}\right)^{1-n}}$ $\eta_0 = D_1 \exp\left(\frac{-A_1(T - T_c)}{A_2 + (T - T_c)}\right)$ $T_c = D_2 + D_3 P$ $A_2 = \tilde{A}_2 + D_3 P$ For PVT model (modified Tait model):

			$\hat{V} = \hat{V}_0[1 - C\ell n(1 + P/B)] + \hat{V}_t$ $\hat{V}_0 = \begin{cases} b_{1S} + b_{2S}\overline{T}, \text{if } T \le T_t \\ b_{1L} + b_{2L}\overline{T}, \text{if } T > T_t \end{cases}$ $B = \begin{cases} b_{3S} \exp(-b_{4S}\overline{T}), \text{if } T \le T_t \\ b_{3L} \exp(-b_{4L}\overline{T}), \text{if } T > T_t \end{cases}$ $\hat{V}_t = \begin{cases} b_7 \exp(b_8\overline{T} - b_9 P), \text{if } T \le T_t \\ 0, \text{if } T > T_t \end{cases}$
2.4	**SIMULATED INPUT**	N/A	

3	**SOLVER AND COMPUTATIONAL TRANSLATION OF THE SPECIFICATIONS**		
3.1	**NUMERICAL SOLVER**	FEM	
3.2	**SOFTWARE TOOL**	Moldex 3d and Moldflow	
3.3	**TIME STEP**	According the requirements. Mesh dependent.	
3.4	**COMPUTATIONAL REPRESENTATION**	**PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL**	Flow pattern with processing parameters
3.5	**COMPUTATIONAL BOUNDARY CONDITIONS**	Geometry of part (specimen).	
3.6	**ADDITIONAL SOLVER PARAMETERS**	Built in solver	

Post processing

The "raw output" calculated by the model is per definition the physics variable in the PE(s).
This is already specified in the entry 2.2 and will appear in your dark green circle in the workflow picture.

This output is often processed by a post processor in order to calculate values for physics variables for different entities that can be input to the next model. Or the output is homogenised for larger volumes in the form of a MR or Descriptor Rule that are the final output of the total simulation.
This will appear in your light green circle in the workflow picture and also in 2.4 of the next model.

The methodology (often including new physics) used to do this calculation is to be documented.

4	**POST PROCESSING**
4.1	**THE PROCESSED OUTPUT**	Flow pattern from liquid to solid.
4.2	**METHODOLOGIES**	Processing through solver viewer
4.3	**MARGIN OF ERROR**	1% (default)