

# Tutorial: Circuit-based Electromagnetic Transient Simulation

Amritanshu Pandey, University of Vermont

December 26, 2022

## Introduction

This document is a tutorial to help students and working professionals learn the inner workings of tools that perform electromagnetic transient (EMT) simulations. The tutorial is introductory in that it shows the workings through two simple examples and expects the reader to build expertise through more detailed readings. Nonetheless, this document is an excellent first step for anyone interested in the subject, especially those interested in building their own tools.

Before we dive into EMT models and simulation, it is pertinent to discuss the two types of commonly performed time-domain simulations in the power community:

- **Transient Stability:** Performs time-domain simulations on balanced networks (only positive sequence components). It does not include network impedance response and models them moving from one steady-state to another instantaneously. Therefore transient stability frameworks model the network constraints with single-frequency power flow equations.
- **Electromagnetic Transient Simulation:** Works exactly as circuit-simulation tools (for instance, SPICE and CADENCE). It models all three phases, including the transient response for the network impedance components.

This document focuses on electromagnetic transient simulation. These are becoming increasingly common (with significant active research) due to the introduction of inverter-based resources on the grid. The document provides a step-by-step tutorial on running an EMT simulation for simple networks from the first principles. It uses concepts from circuit simulation, Newton-Raphson, and numerical integration with difference methods. Good references to brush up on these topics are [1],[2], and [3].

Furthermore, this document uses modified nodal analysis (MNA) to encapsulate network physics. Alternate approaches like Tree Link Analysis (TLA) and Sparse Tableau Analysis (STA) [1] exist but are not covered in this document. In naive nodal analysis, the currents at each node are summed to zero (i.e., Kirchhoff's current law) to satisfy network physics. However, the method fails when the network includes voltage sources, as the current through the voltage source is not implicitly known. Therefore, to include voltage sources, the modified version adds one additional constraint per voltage source to the set of nodal equations. These additional constraints give us the currents through the voltage sources as new variables. The term *modified* in MNA is to refer to this fact. Finally, the readers must note that anytime in this document when they come across the following terms: system matrix,  $Y$  matrix, solution matrix, or simple nodal matrix, these all refer to the same thing: a set of MNA equations for the linearized network that is being evaluated at a given time-step  $t$ .

# 1 Tutorial: Electromagnetic Transient Simulation Methodology

We will use an equivalent circuit framework [4], [5] for power grids to develop EMT models for two simple toy networks. These toy examples are based on a simple 2-node three-phase power network with different load configurations. We consider the following load scenarios:

- a linear wye-connected series RL load (see Fig. 3)
- a non-linear induction motor load (see Fig. 11)

Note: For both examples, we assume that the load is connected in a WYE configuration and all the neutrals in the network are grounded.

The illustration of the two-node power network with a generic load model is shown in Fig. 1. It includes power system verbiage such as infinite bus and wye-connected load. For the moment, the infinite bus (on node 1) can be thought of as the power source, and the load bus (on node 2) can be thought of as the power sink.

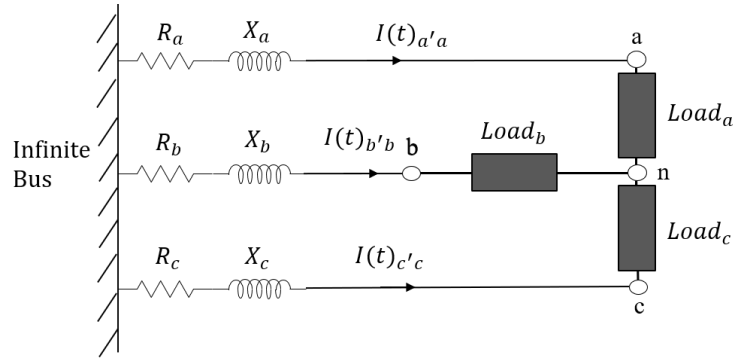


Figure 1: Power system representation of the 2-bus power network.

The illustration in Fig. 1 is a power system representation. To perform EMT simulation with an equivalent circuit framework, as a first step, we represent any power grid-specific elements as electric circuit elements. The network in Fig. 1 has three components i) infinite bus, ii) transmission line (a very simplified version), and iii) load. Two of these three components (infinite bus and transmission line) are shared across the two examples. Therefore, we begin with them. The infinite bus is trivial to model as a circuit element. It is simply an ideal voltage source with a known magnitude and phase angle. For a three-phase network, the infinite bus is modeled by a set of three independent voltage sources (offset by  $120^\circ$ ) connected in a wye formation. The network with an infinite bus replaced with circuit elements is shown in Fig. 2. The transmission line model with R and X values is already represented as a set of circuit elements and does not require translation.

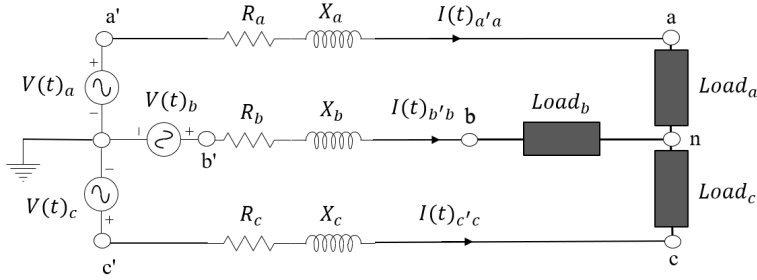


Figure 2: Power system representation of the 2-bus power network.

The last step is to convert the load component in Fig. 2 into an equivalent circuit before getting started with EMT-related steps. Loads in power systems correspond to various devices ranging from large induction motors to resistive heating loads. We will translate the two load models into equivalent circuits in their respective sub-sections.

### 1.1 Wye-connected RL Load

The first example represents the load via a wye-connected series RL impedance. With series RL impedance as the load component, the equivalent circuit representation of the 2-bus network is trivial and is shown in Fig. 3. Simple observation confirms that all elements in this equivalent circuit are linear; therefore, to obtain the transient response, we only need to *recursively* solve the network as a function of time. *Iterative* solves with Newton-Raphson (NR) are only necessary in presence of nonlinear components. We will learn about that in the induction motor (IM) example.

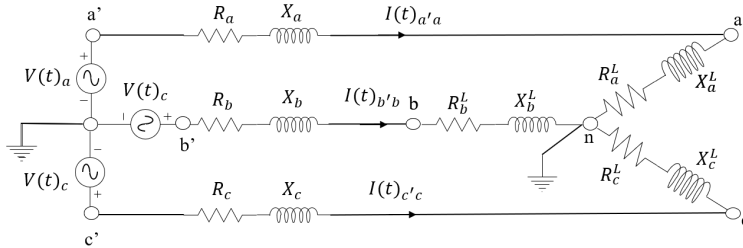


Figure 3: Power system representation of the 2-bus power network.

In current EMT tools (for instance, EMTP-RV or PSCAD), a sequence of steps that are followed to obtain the time-domain response of a linear power network are as follows:

1. First, ordinary differential equations (ODEs) corresponding to the sum of currents at each node in the network are defined
2. Second, numerical integration approximation (such as trapezoidal or backward Euler) is applied to these ODE equations to convert them from differential algebraic equation (DAE) form to purely algebraic equation (AE) form

3. Third, if the network is linear, the final solution for each time instance is obtained via a linear solve of the algebraic equations for each time-tick

Note that the equations for subsequent time steps are coupled through terms obtained from numerical integration. So currents at  $t = n + 1$  will be functions of currents at  $t = n$ . We will see this in more detail while constructing the companion models in the following subsections. Also, note that linear equations are recursively solved to traverse through simulation time by choosing a **time-step** ( $\Delta t$ ). The value of this time step depends on the local truncation error (LTE). More details on the calculation of LTE can be found in [1].

Next, I will describe how to tackle the same problem in an equivalent circuit approach (ECA). To obtain the time-domain response of the network with ECA, we apply **two (2) circuit-simulation tricks** to enable efficient construction of the linearized nodal equations ( $Y_t V_t - J_t = 0$ ) for each time step ( $t$ ) in the transient simulation.

**First, rather than** defining the ordinary differential equation (ODE) that captures the KCL constraint (sum of currents) at each node, we construct specialized symbolic stamps for each component in the network (e.g., resistance, inductance, etc.). Note that a stamp(s) here refers to term(s) that are to be added to the solution matrix. We use these stamps to implicitly construct the system matrix at each time tick. To do so, we parse through each circuit element in the network (for each time tick), and we add the stamps corresponding to this element in the solution matrix. Once completed, the matrix (completed by adding stamps from each component) will be equivalent to if we constructed the matrix by adding linearized nodal equations in each row. In this approach, rather than arduously constructing nodal equations for each node (imagine a circuit with million nodes), we only have to symbolically construct stamps for the number of types of circuit elements. Once that is done, all we need to do is add the stamps for each circuit element into the solution matrix.

**Second circuit simulation trick** corresponds to the numerical integration of differential terms. Instead of applying numerical integration to all nodal ODE equations, we develop and use companion circuits for time derivative elements (e.g., inductance and capacitance) in the network.

In ECA, with the two tricks, we follow the following steps sequentially to enable modular construction and solution of the system matrix at each time tick without ever having to define the complete set of nodal equations explicitly:

1. Represent time-derivative elements in the circuit (e.g., inductor, capacitor etc.) with their companion circuits
2. Develop a specialized stamp for each element in the circuit (with derivative elements replaced by their companion circuits)
3.  $\forall t \in \tau = \{t_0, \dots, t_{final}\}$ , parse through the overall network (generally a netlist or input file of sorts) and stamp the terms corresponding to each element in a sparse linear matrix (remember at this point, time-derivative terms are replaced by their companion models)
4. Solve the linear problem  $Y_t V_t = J_t$ , to obtain the solution of state variables at time  $t$

This approach allows us to implicitly obtain and solve the linearized nodal equations without constructing them symbolically. This is highly beneficial for developing heuristics and for solving large-scale networks. More on that later.

### 1.1.1 Companion Model for Inductors and Trapezoidal Rule

Returning to the RL-load example, the inductor is the only element with a time derivative term in this network (see Fig. 3). Therefore all we have to do is construct a companion model for the inductor. We then replace all the inductors in the network with the companion model. We now discuss the construction of companion models.

This tutorial will use Trapezoidal integration approximation to construct companion models. Other Euler-based or higher-order numerical integration methods can also be used following the same approach. Applying the trapezoidal integration approximation, the companion circuit for the inductor can be derived as follows (for more details, read [1]):

$$L \frac{di}{dt} = v \quad (1)$$

$$di = \frac{v dt}{L} \quad (2)$$

$$\int_{t_n}^{t_{n+1}} di = \int_{t_n}^{t_{n+1}} \frac{v dt}{L} \quad (3)$$

$$(i(t_{n+1}) - i(t_n)) = (v(t_{n+1}) + v(t_n)) \frac{\Delta t}{2L} \quad (4)$$

$$i(t_{n+1}) = i(t_n) + v(t_n) \frac{\Delta t}{2L} + v(t_{n+1}) \frac{\Delta t}{2L} \quad (5)$$

The equation in (5) has three terms. The last term  $\frac{\Delta t}{2L} v(t_{n+1})$  describes the relationship between current and voltage at time  $t_{n+1}$  and is a conductance  $G$ . The first two terms are constants that are known at prior time-point  $t_n$  states and therefore are represented via a constant current source  $I$ . Aggregating these elements into a circuit, we get the equivalent circuit in Fig. 4, and it is a companion model for an inductor with trapezoidal numerical integration approximation.

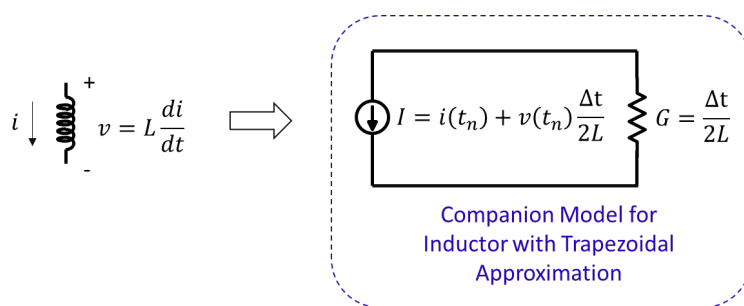


Figure 4: Inductor Trapezoidal Companion Circuit. [Reconstructed from Pileggi, Carnegie Mellon ECE 18-762 Notes]

With the companion model constructed, we can replace the inductor elements in Fig. 3 with the respective companion circuits. Following this step, we obtain a time-dependent equivalent circuit (see Fig. 5), which can be characterized entirely by linear algebraic equations. This set of equations is repeatedly solved to find the solution to the electromagnetic transient response of the network.

However, as discussed, the trick is to avoid explicitly constructing the complete equivalent circuit and corresponding nodal equations ( $YV - J$ ) for each node. Instead, we will follow a modular approach. As we encounter each element in the netlist of the network, we will add the

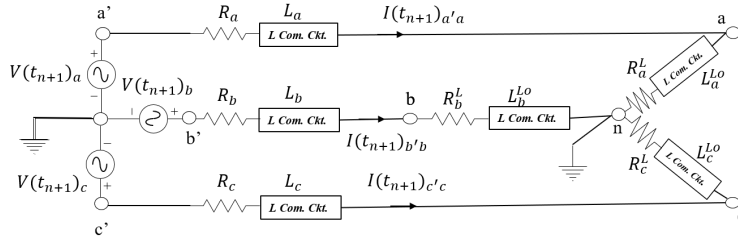


Figure 5: Equivalent circuit with companion models.

specialized stamps corresponding to that element into the sparse system matrix. In this example (see Fig. 5), there are three ideal voltage sources, six resistance and six inductor models. Therefore, we will cover the stamps for these elements, which will be sufficient to construct the linearized nodal equations for each time-step ( $Y_t V_t - J_t = 0$ ) for the network in Fig. 5. By recursively solving these equations over time from the initial time-step  $t_0$  to the final time step  $t_{final}$ , we can obtain the time-domain response of this network. Next, we will cover the stamps for these elements.

Note: We will refer to the sparse linear matrix for the system as  $Y$  from here on. Each time step will have a different system matrix notated by  $Y_t$  or sometimes simply by  $Y$ . Further, even for each time step, the matrix can be expressed as the sum of two separate matrices:  $Y_t = Y_t^{lin} + Y_t^{nlin}$ . Terms in  $Y_t^{nlin}$  change for each iteration of NR, whereas the  $Y_t^{lin}$ , remain fixed. Because this circuit is linear, we do not have  $Y_t^{nlin}$ .

### 1.1.2 Stamp for resistors

A resistance  $R$  between node  $m$  and  $n$  has the following stamps in the nodal matrix  $Y$ :

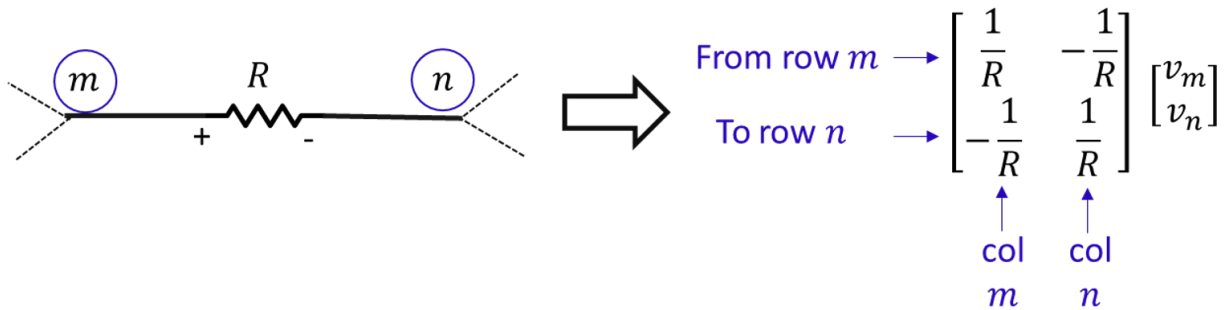


Figure 6: Resistance stamps in the nodal matrix. [Redrawn from CMU, ECE, 18-762 Notes]

The general approach is that when you encounter resistance in an input file, add the terms defined in Fig. 6 in sparse linear matrix  $Y$  corresponding in the index locations:  $(m, m)$ ,  $(m, n)$ ,  $(n, m)$ , and  $(n, n)$ . Here node  $m$  is the from node of the resistor, and  $n$  is the to node of the resistor.

### 1.1.3 Stamps for an independent current source

An independent current source stamp  $I$  between node  $m$  and  $n$  will add no additional stamps to matrix  $Y$  but instead will add terms to column vector  $J$ . The column vector  $J$  only consists of constant terms that are not a function of any state in the network. The negative sign in the  $J$

vector is due to moving the constant number to the RHS of the equation. However, keep in mind the direction of the current flow from the independent current source for the signage.

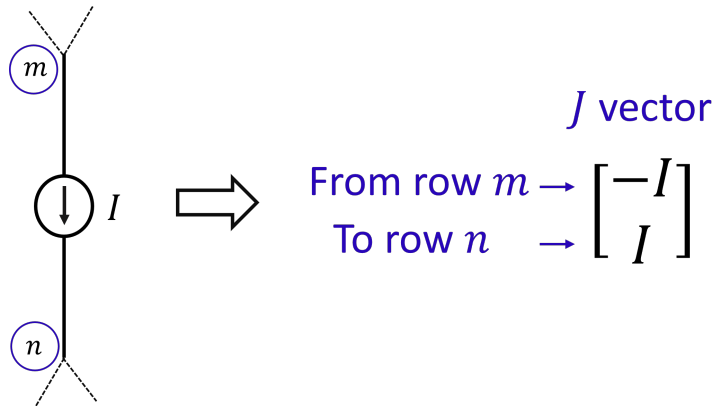


Figure 7: Independent current source stamps in  $J$  vector. [redrawn from Pileggi, CMU 18-762 Notes]

### 1.1.4 Stamps for inductors

Inductor consists of time-derivative terms. Therefore, as a first step, each time we encounter an inductor in the netlist between node  $m$  and node  $n$ , we will have to stamp the corresponding companion circuit. Remember doing so will replace the time-derivative terms with algebraic expressions. It consists of a current source and a resistor in parallel between nodes  $m$  and  $n$  as seen in Fig. 4. As we have already covered stamps for resistance and current source, follow the steps in the previous subsections for those components. The key thing to note is that stamp values for inductors in system matrix  $Y$  will change for each time step and must be updated accordingly. This is because the magnitude of the independent current source in the companion circuit will be a function of currents and voltages from the last time tick  $t_n$ .

### 1.1.5 Stamps for independent voltage source

Next, we derive the stamps for a voltage source. To include an independent voltage source between nodes  $m$  and  $n$ , nodal equations are augmented to include an additional row. With an extra row comes an additional variable representing the magnitude of current flow through the voltage source. The term *modified* in modified nodal analysis (MNA) is to notate this modification. The additional row (for each voltage source) constrains the potential difference between node  $m$  and node  $n$ :

$$V_m - V_n = V_S \quad (6)$$

, and the additional current is notated by  $(i)$ . The stamps in the nodal matrix ( $Y$ ) and source vector ( $J$ ) for independent voltage source are shown in Fig. 8

Now that we have learned how to develop stamps for the resistor, inductor, current source, and voltage source, we can construct the nodal matrix ( $Y$ ) and source vector ( $J$ ) for the simple power system network in Fig. 3 for each time  $t_{n+1}$ . To obtain the time-domain response of the network, the set of linearized equations is recursively solved with a sparse matrix solver (using LU factorization). Note that to obtain the network's response at time  $t = 0^+$ , we require the network states at time  $t_n = 0$ . For this particular network, these are needed to initiate the terms in inductor stamps dependent on the prior time step. Not just that, it also helps dictate the system state at which the time-domain response begins.

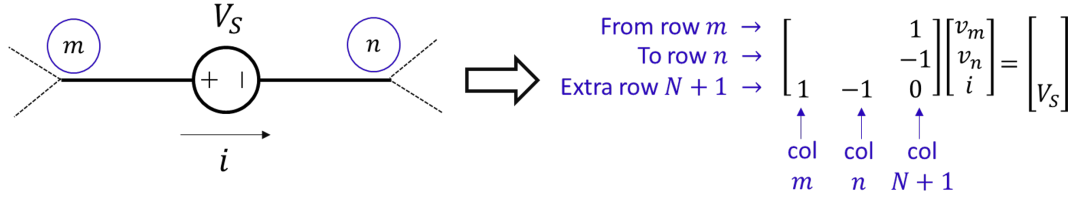


Figure 8: Independent voltage source stamps. [redrawn from Pileggi, CMU- 18-762 Notes]

### 1.1.6 Initialization

We can initialize the linear circuit through AC or DC analysis. In DC analysis, we replace the sinusoidal voltage sources with DC voltage sources. In AC analysis, we convert the inductors to an impedance at a single frequency.

Here we will describe the DC analysis approach to initialization. Let us assume that the network is in the DC state between  $t = -\infty$  and  $t = 0$ . At  $t=0$ , it switches to the AC sinusoidal source, and let us assume the switching is smooth (i.e., DC voltage and AC voltage at time  $t = 0$  are identical). To know the initial states of the circuit at  $t = 0$ , we perform DC analysis, wherein the following condition holds true for memory elements (inductors and capacitors):

$$L \frac{di}{dt} = 0 \quad @DC \quad (7)$$

$$C \frac{dv}{dt} = 0 \quad @DC \quad (8)$$

From these conditions in (7) and (8), we can infer that during the DC state, the inductor is short-circuited ( $V_L = 0$ ) and the capacitor is open-circuited ( $I_C = 0$ ). For this example, the DC equivalent of the network in Fig. 5 is shown via a network in Fig. 9. With access to DC equivalent circuit, we stamp the elements in the circuit into the system matrix  $Y_{dc}$  and vector  $J_{dc}$  and solve them using a sparse linear solver. The solution is used to define the system state at  $t = 0$ .

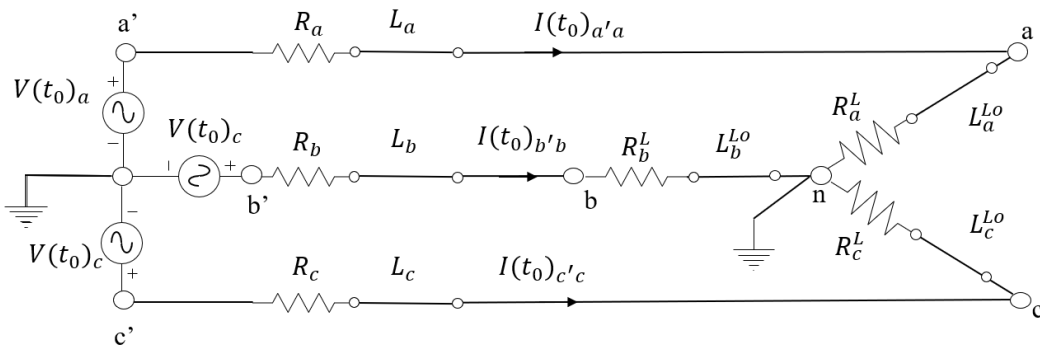


Figure 9: Corresponding equivalent circuit during DC state. Note that the sinusoidal voltage sources are set to a fixed DC value in the case of DC analysis.



### 1.1.7 Time-domain solution

With access to the initial state at  $t = 0$ , we recursively solve a system matrix (moving forward by  $\Delta t$  in each recursion) following steps described in Fig. 10 to obtain the time-domain response from  $t = 0$  to  $t = t_{final}$ . We dynamically adjust  $\Delta t$  at each step based on the trade-off between simulation run-time and local truncation error (LTE).

One facet that this example does not cover is how to handle nonlinear terms. We will learn how to include nonlinear terms in the following example with IM as an electric load. We will also learn to use an alternative form (other than MNA) to describe the system's behavior. In addition to the electrical physics of IM, we will incorporate mechanical physics, like rotor speed, with the circuit-theoretic paradigm, into the mathematical formulation.

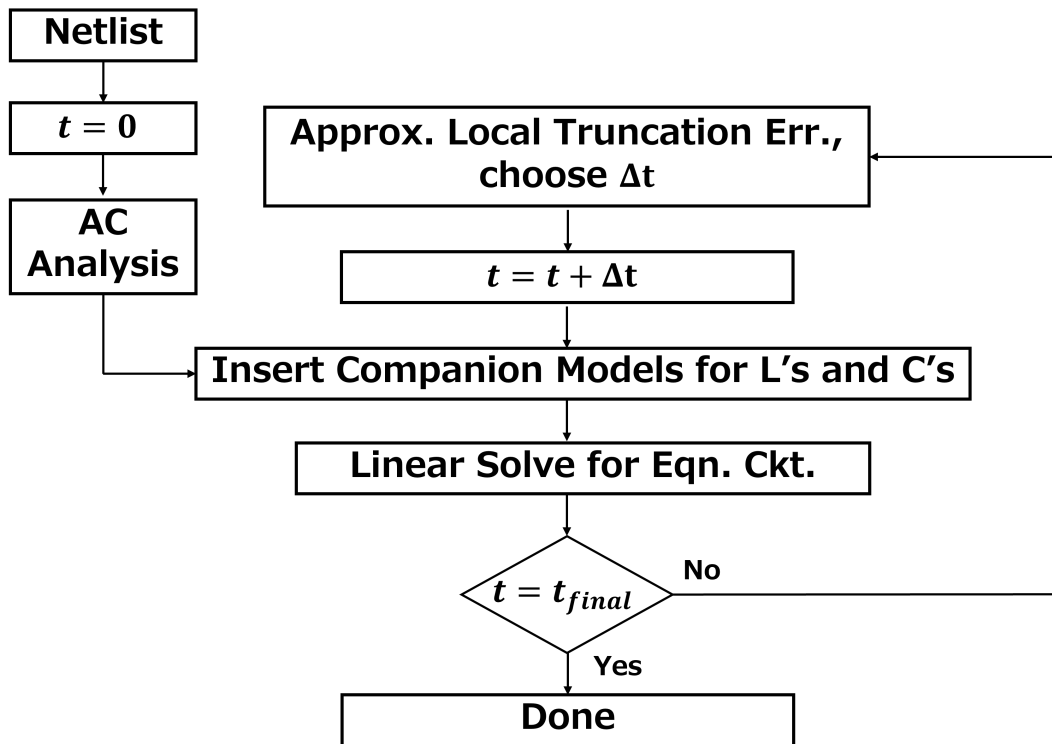


Figure 10: Running EMT simulation for linear networks.

## 1.2 Induction Motor Load

In the second example, we will learn how to obtain the time-domain response for a non-linear grid component: the induction motor (IM). To construct a toy power grid network with IM, we will replace the RL load in the first example with an IM load (see Fig. 11).

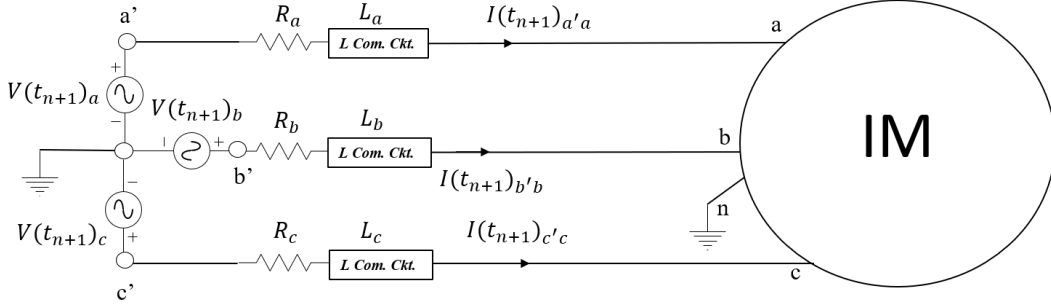


Figure 11: 2-bus network with induction motor load.

Analyzing IM in the  $abc$  frame can be difficult. The flux generated by the three-phase IM in  $abc$  frame has time-varying coefficients in its voltage terms due to the sinusoidal nature of the mutual inductance. This makes the analysis of three-phase IM cumbersome in the  $abc$  reference frame. However, this undesirable feature can be eliminated by the use of  $dq$  transformation.  $dq$  transformation is a linear transformation. It can be performed by choosing one of the three reference frames: i) synchronous reference frame, ii) stationary reference frame, and iii) rotating reference frame.

For derivations in this document, we will use a stationary reference frame.  $abc$  variables (both currents and voltages) are converted to  $dq$  variables by following matrix transformation:

$$[F_{0dq}] = [P_\theta][F_{abc}] \quad (9)$$

where:

$$[P_\theta] = \frac{2}{3} \begin{bmatrix} 0.5 & 0.5 & 0.5 \\ \cos(\theta) & \cos(\theta - \lambda) & \cos(\theta + \lambda) \\ \sin(\theta) & \sin(\theta - \lambda) & \sin(\theta + \lambda) \end{bmatrix} \quad (10)$$

$\lambda$  represents the phase difference between phases  $abc$  and is:

$$\lambda = \frac{2\pi}{3} \text{rad} \quad (11)$$

After applying  $dq$  transformation to IM and decoupling  $abc$  components from  $dq$  components using controlled current and voltage sources, the network model in Fig. 11 is modified to one in Fig. 12.

Note that controlled current and voltage sources in Fig. 12 encapsulate the math behind  $dq$  and inverse  $dq$  transformation for the variables.  $dq$  transformation is applied to capture  $abc$  network voltages and supply those to IM in  $dq$  frame. Inverse  $dq$  transformation is used to convert the  $dq$  currents through the IM into  $abc$  frame such that they can be fed into the network.

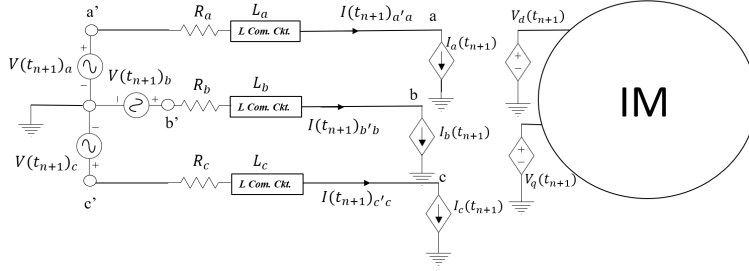


Figure 12: 2-bus network with induction motor load after dq transformation.

Fig. 12 has three components: i) the infinite bus represented by a set of wye-connected independent voltage sources, ii) the transmission line represented by a series RL circuit, and iii) an IM. We have covered how to develop stamps and companion circuits for infinite bus and transmission line elements. Now, we will learn how to represent the physics of an IM with an equivalent circuit. We will also learn how to replace the time derivative terms in the IM equivalent circuit with an equivalent companion circuit.

For stationary  $dq$  frame, the equations that represent the electrical component of IM physics are given by:

$$v_{ds} = R_s I_{ds} + p\psi_{ds} - \psi_{qs} p\theta \quad (12)$$

$$v_{qs} = R_s I_{qs} + p\psi_{qs} + \psi_{ds} p\theta \quad (13)$$

$$v_{dr} = R_r I_{dr} + p\psi_{dr} - \psi_{qr} p\beta \quad (14)$$

$$v_{qr} = R_r I_{qr} + p\psi_{qr} + \psi_{dr} p\beta \quad (15)$$

where  $p$  is the differential operator and,

$$\psi_{ds} = (L_{ls} + L_m) I_{ds} + L_m I_{dr} \quad (16)$$

$$\psi_{dr} = (L_{lr} + L_m) I_{dr} + L_m I_{ds} \quad (17)$$

$$\psi_{qs} = (L_{ls} + L_m) I_{qs} + L_m I_{qr} \quad (18)$$

$$\psi_{qr} = (L_{lr} + L_m) I_{qr} + L_m I_{qs} \quad (19)$$

and  $\theta = 0$ ,  $\beta = -\theta_r$  and  $p\beta = \omega_r$ .

In the IM equations,  $L_{ls}$  and  $L_{lr}$  represent the leakage inductance of the stator circuit and rotor circuit, respectively.  $L_m$  is the mutual inductance between the rotor and stator circuits.  $R_s$  and  $R_r$  are the stator and rotor resistance, respectively. The nonlinearity in the electrical part of the IM is due to the speed voltage terms ( $\psi_{qr}\omega_r$ ,  $\psi_{dr}\omega_r$ ). In addition to the equations for the IM electrical part given by (13)-(19), the mechanical part of the IM is defined by the swing equation, a single differential equation (20):

$$p\omega_r = \frac{(T_E - T_L - D\omega_r)}{J} \quad (20)$$

where, electrical torque  $T_E$  is described by an interaction between the IM currents, which also introduces nonlinearities in the IM model:

$$T_E = \frac{3}{4}L_m N_p (I_{dr} I_{qs} - I_{qr} I_{ds}) \quad (21)$$

$T_E$  is the electrical torque of the IM in  $N.m$  and  $J$  is the motor net inertia in  $kg.m^2$ .  $N_p$  is the number of poles in the IM. The load torque  $T_L$  is generally described as a polynomial function of rotor speed.

Mapping the IM into an equivalent circuit with  $\omega_s$  set to 0 in the stationary frame, we get the modified equivalent circuit shown in Fig. 13, with the following additional terms and constraints:

$$L_s = Ll_s + L_m \quad (22)$$

$$L_r = Ll_r + L_m \quad (23)$$

$$V_{dr} = 0 \quad (24)$$

$$V_{qr} = 0 \quad (25)$$

Note that the rotor voltages  $V_{dr}$  and  $V_{qr}$  are shorted in most IM designs.

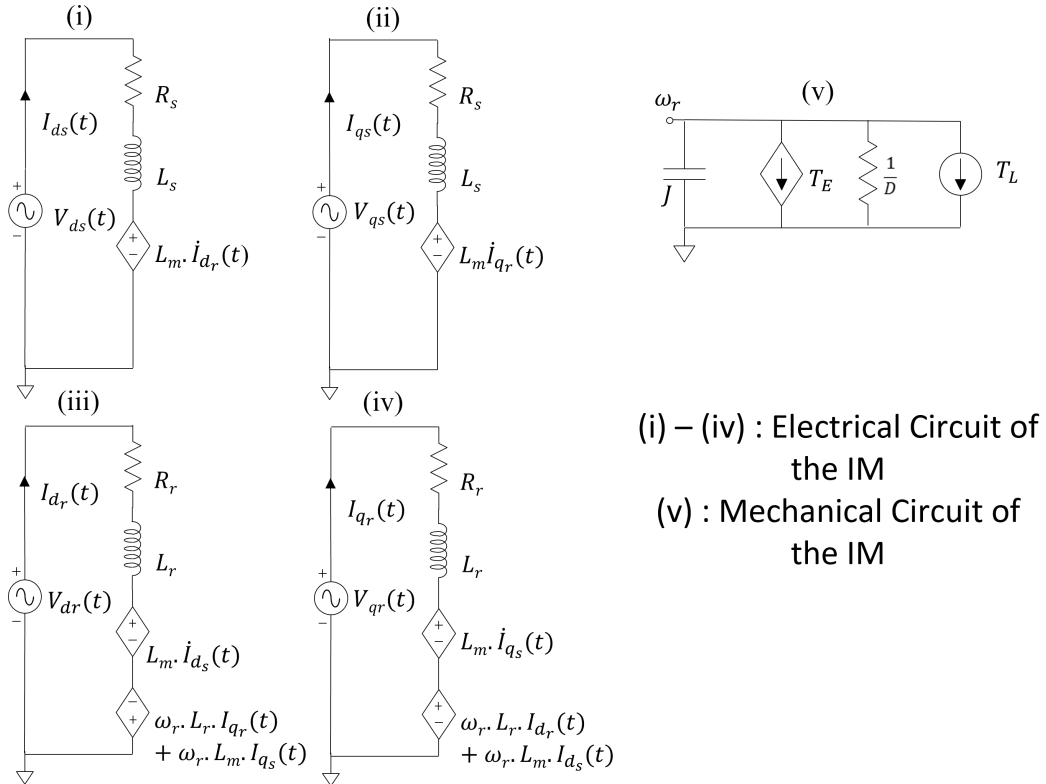


Figure 13: Equivalent circuit for IM in the stationary frame.

One can observe in Fig. 13 that aside from self-inductance ( $L_s$  and  $L_r$ ), the IM equivalent circuit also includes time-derivative terms (given by  $p$ ) for mutual inductance ( $L_m$ ) and rotor speed ( $\omega_r$ ). We have previously learned how to approximate time-derivative terms for self-inductance by constructing and stamping the corresponding companion circuits. Here, we will derive the companion circuits for combined self and mutual inductance elements (see blue elements in Fig. 14). We will use a simple two-coil example in Fig. 15 to develop the companion circuit for combined self- and mutual inductance. Later in the section, we will learn how to construct the companion circuit for time-derivative terms corresponding to rotor speed.

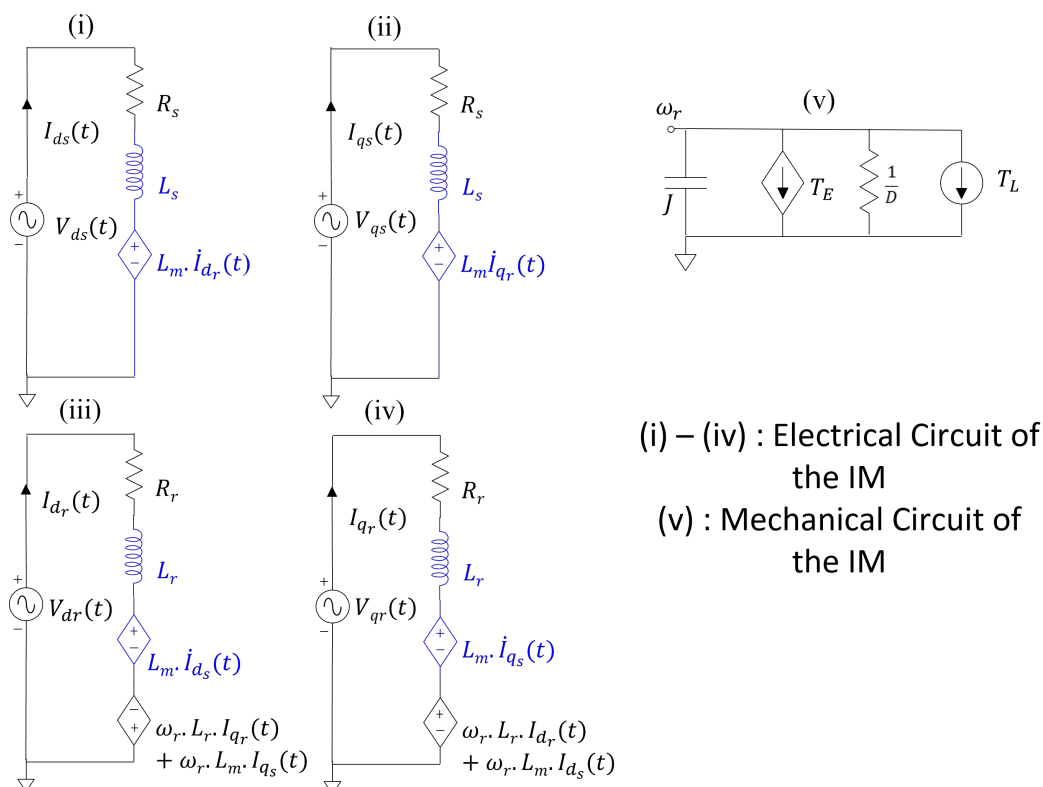


Figure 14: Illustration of combined self and mutual inductances in IM.

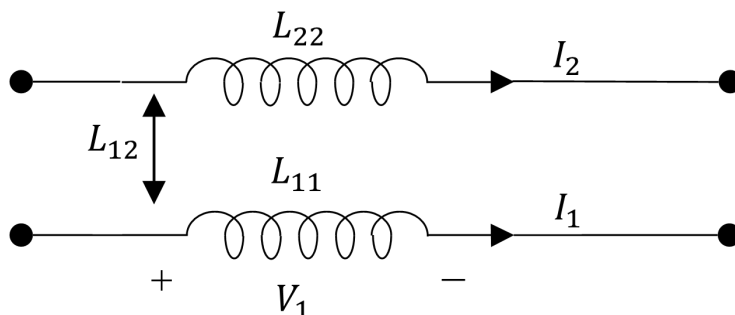


Figure 15: Two coil example.

We construct the companion circuit for combined self and mutual inductances to approximate its time-derivative behavior by a set of algebraic equations. This way, we can approximately solve a

set of ODEs parameterized by self- and mutual- inductances by recursively solving a set of algebraic equations over time. We begin the derivation by observing a two-coil example in Fig. 15, which has two pairs of self- and mutual-inductance (self:  $L_{11}$  or  $L_{22}$ , mutual:  $L_{12}$ ). We focus on voltage  $V_1$  expression in the first coil. It can be represented as a sum of the voltage across the self and mutual inductances:

$$V_1(t) = L_{11} \frac{dI_1}{dt} + L_{12} \frac{dI_2}{dt} \quad (26)$$

The voltage expression in (26) is an ODE that includes time-derivative terms. Therefore, we apply the trapezoidal integral rule to approximate the ODE's solution. With the trapezoidal integral rule applied, the voltage across self- and mutual- inductance (26) can be represented by the sum of the following difference equations, which are purely algebraic terms:

$$V_{11}(t_{n+1}) = \frac{2L_{11}}{\Delta t} (I_1(t_{n+1}) - I_1(t_n)) - V_{11}(t_n) \quad (27)$$

$$V_{12}(t_{n+1}) = \frac{2L_{12}}{\Delta t} (I_2(t_{n+1}) - I_2(t_n)) - V_{12}(t_n) \quad (28)$$

Therefore the total voltage induced in coil 1 ( $V_1(t) = V_{11}(t) + V_{12}(t)$ ) is given by:

$$V_1(t_{n+1}) = \frac{2L_{11}}{\Delta t} (I_1(t_{n+1}) - I_1(t_n)) - V_{11}(t_n) + \frac{2L_{12}}{\Delta t} (I_2(t_{n+1}) - I_2(t_n)) - V_{12}(t_n) \quad (29)$$

and after re-arranging the terms:

$$V_1(t_{n+1}) = \frac{2L_{11}}{\Delta t} I_1(t_{n+1}) + \frac{2L_{12}}{\Delta t} I_2(t_{n+1}) - \left( \frac{2L_{11}}{\Delta t} I_1(t_n) + \frac{2L_{12}}{\Delta t} I_2(t_n) + V_{12}(t_n) + V_{11}(t_n) \right) \quad (30)$$

The voltage at time  $t_{n+1}$  across the coil 1 is approximated using (30) and is represented as an equivalent circuit in Fig. 16. The **third term** is only dependent on the historical values of the variables and can be represented by an **independent voltage source**  $V_{1.hist}$ . Similarly, a **resistance**  $R_{EQ}$  can represent the **first term** as it represents a linear relationship between the voltage and current and a **current controlled voltage source** can represent the **second term**.

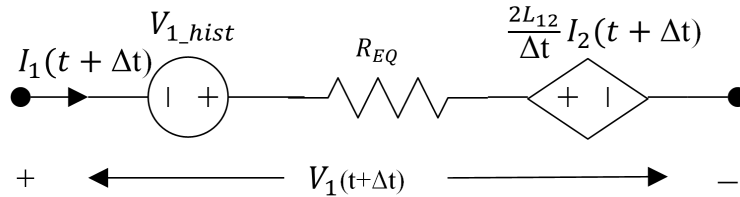


Figure 16: Two coil example companion circuit.

By replacing the time-derivative terms for self- and mutual-inductance with the corresponding companion circuit in Fig. 16, we can solve the set of ODEs corresponding to the electrical part of IM over a time duration with NR. But before, we must learn how to address the time-derivative

term in the mechanical part of the IM model. The variables in the mechanical part impact the electrical part and vice-versa. Therefore we need to solve the mechanical part concurrently with the electrical portion of IM. The approach for solving the mechanical part is trivial. On close observation, one can see the mechanical circuit of the IM is analogous to a parallel RC electrical circuit with current sources, where inertia  $J$  is analogous to capacitance  $C$ ,  $T_L$  is analogous to a constant current source  $I$ ,  $D$  is analogous to a conductance  $G$ ,  $T_E$  is analogous to a current controlled current source (see (21)), and  $\omega_r$  is analogous to the voltage at the mechanical the circuit. The only time-derivative term in the mechanical circuit corresponds to the partial derivative of rotor speed times inertia ( $J \frac{d\omega_r}{dt}$ ), which is analogous to a current through a capacitor. Therefore, we will learn how to construct a companion circuit for a capacitor and replace the time-derivative term in the mechanical part with the analogous companion circuit. Once we have replaced the time-derivative term, we can represent the physics of the mechanical part of IM with a single nodal equation, which can be iteratively solved for a single time step along with electrical equations using NR.

To construct the companion model for a capacitor, we will use Trapezoidal integration approximation following the same approach we applied in the case of inductors. Applying the trapezoidal integration approximation, the expression for current/voltage through/across a capacitor at time  $t_{n+1}$  given values at time  $t_n$  can be expressed as follows:

$$C \frac{dv}{dt} = i \quad (31)$$

$$dv = \frac{idt}{C} \quad (32)$$

$$\int_{t_n}^{t_{n+1}} dv = \int_{t_n}^{t_{n+1}} \frac{idt}{C} \quad (33)$$

$$(v(t_{n+1}) - v(t_n)) = (i(t_{n+1}) + i(t_n)) \frac{\Delta t}{2C} \quad (34)$$

$$v(t_{n+1}) = v(t_n) + i(t_n) \frac{\Delta t}{2C} + i(t_{n+1}) \frac{\Delta t}{2C} \quad (35)$$

The equation in (35) has three terms. The last term  $\frac{\Delta t}{2C}$  describes the relationship between current and voltage at time  $t_{n+1}$  and is a conductance  $G$ . The first two terms are constants that are known from prior time-point  $t_n$  and therefore are represented via a constant current source  $I$ . Aggregating these elements into a circuit, we get the equivalent circuit in Fig. 4. This is a companion model for a capacitor with trapezoidal numerical integration approximation. In the case of IM, note that we must have analogous symbols like inertia  $J$  instead of capacitor  $C$  and rotor speed  $\omega_r$  instead of voltage  $v$ .

### 1.2.1 Handling nonlinear terms

In the case of IM, we began with a set of models characterized by a set of differential-algebraic equations (DAEs), which had both differential and nonlinear terms. As a first step, we used companion circuits to approximate the differential term behavior using a set of algebraic terms.

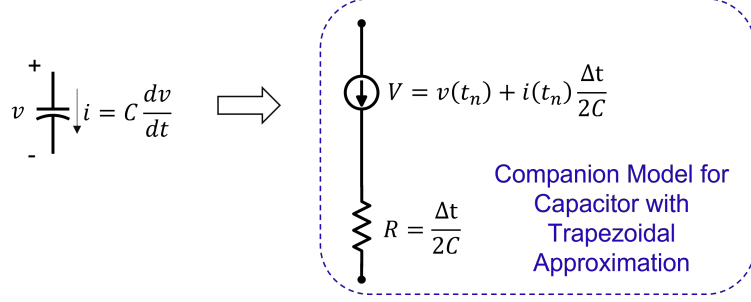


Figure 17: Capacitor Trapezoidal Companion Circuit. [redrawn from Pileggi, 18-762 Notes]

Next, we must linearize the nonlinear terms and iteratively solve the equations using a sparse linear solver.

We linearize the nonlinear terms using first-order Taylor's approximation, similar to the case of nonlinear power flow analysis. However, instead of linearizing the nodal or loop equations, we linearize the nonlinear terms corresponding to each component and develop a linearized equivalent circuit. Then for each iteration of NR, we iteratively update the stamps corresponding to this component in the system matrix  $Y$  and resolve.

Let us write out the general use of first-order Taylor approximation. For general nonlinear function  $f(x)$  ( $x : \mathbb{R}^{N \times 1}$ ), the first order Taylor approximation is given by:

$$f^{k+1}(x) = f^k(x) + f'(x)^T (x^{k+1} - x^k) \quad (36)$$

In the IM model in Fig. 13, for the electrical part, the non-linearity stems from four speed-voltage terms ( $\omega_r L_r I_{dr}$ ,  $\omega_r L_m I_{ds}$ ,  $\omega_r L_r I_{qr}$ , and  $\omega_r L_m I_{qs}$ ). For further discussion consider one amongst them:  $f(\omega_r, I_{dr}) = \omega_r L_r I_{dr}$ . This expression is a nonlinear function of the rotor speed ( $\omega_r$ ) and direct-axis rotor current ( $I_{dr}$ ). The linearized approximation for this expression used in  $(k + 1)^{th}$  NR iteration is:

$$f^{k+1}(\omega_r, I_{dr}) = f^k(\omega_r, I_{dr}) + \left( \frac{\partial f}{\partial \omega_r} \right)_k (\omega_r^{k+1} - \omega_r^k) + \left( \frac{\partial f}{\partial I_{dr}} \right)_k (I_{dr}^{k+1} - I_{dr}^k) \quad (37)$$

Similar to this nonlinear term, other nonlinear terms in the IM are also linearized and stored in symbolic form. These terms are added (or stamped) in the system matrix  $Y$  and updated for each iteration of NR.

### 1.2.2 Stamping IM equations

So far, we have discussed how to address nonlinear and differential terms at a component level. The next step is to learn how to add (or stamp) the terms corresponding to various component-based equivalent circuits into a system matrix (such that system-level physics is satisfied). We must also learn when to update these terms to obtain a time-domain solution for the overall circuit. We can stamp most components (except for IM) in the network into the system matrix following the MNA approach as described in Example 1. For IM, we will use an alternative approach. If we were to stamp the IM circuit elements using MNA, we would end up with  $\sim 20$  additional nodes and corresponding equations for each IM we encounter in the network (refer to Fig. 13). Therefore, to reduce the dimension of IM equations, in this tutorial, we will instead use a combination of



Kirchhoff Voltage Law (KVL)-based constraints and KCL-based nodal constraints to stamp the IM model's equivalent circuit elements. With this approach, we will stamp the equations corresponding to electrical sub-circuits (i through iv in Fig. 13) in 4 new rows of the system matrix  $Y$  following KVL-based loop constraints. Specifically, for each electrical sub-circuit in Fig. 13, moving from left to right, we will add stamps (for voltages) for each element such that the net sum of voltages in the loop is equal to zero. For the mechanical part (no. (v) in Fig. 13), we will add terms to matrix  $Y$  for various elements (including linearized  $T_E$  and companion circuits  $\frac{\partial \omega_r}{\partial t}$ ) following KCL-based nodal constraint. With this approach, we only add five new variables and corresponding rows for each instance of IM (instead of  $\omega_r$ ). The variables are  $I_{ds}, I_{qs}, I_{dr}, I_{qr}$ , and  $\omega_r$ .

Note that before stamping the terms into the system matrix  $Y$ , we have to perform two steps. First, we will replace the circuit elements with time-derivative terms with corresponding equivalent companion circuits. Second, we will replace the circuits with nonlinear terms with their linearized approximation. Next, to obtain the transient response of the nonlinear IM network, we will recursively solve the system matrix  $Y$  over time, and we will perform iterations to solve the nonlinear terms. We will update the linearized terms in each iteration. We will update the terms in companion circuits only when we recursively move forward in time.

### 1.2.3 Initialization

For initializing the IM network, we can use many approaches. We will discuss one such approach. Remember because of dq-transformation on IM variables, two sub-circuits evolve (see left and right of Fig. 12). The sub-circuit on the left without the IM equations is a linear circuit. AC analysis can be used to obtain the initial condition for these, assuming a rated complex current draw by IM (which models the  $I_a, I_b$ , and  $I_c$  in Fig. 12). A good guess for a complex current draw by IM can be obtained by running power flow with IM modeled as a PQ load and calculating the current from the solution ( $I = S_{IM}^*/V_{IM}^*$ ). Initialization of the IM components requires handling the nonlinearities due to speed-flux terms. With the proper choice of the reference frame in dq-transformation (rotating reference frame), we can use DC analysis to obtain the initial conditions for the IM circuit. The  $V_{ds}$ , and  $V_{qs}$  voltages are DC-values in the rotating reference frame. We can short the inductor and open the capacitors to obtain the steady-state initial conditions with the source voltages as DC values. The rated voltage at the IM terminals for dq-transformation can be obtained from the power flow solution  $V_{IM}$ . In reality, with a slightly more complex procedure, we can get an exact initial condition for the circuit by solving a set of nonlinear equations representing the overall circuit iteratively using NR.

### 1.2.4 Nonlinear time-domain simulation

For the nonlinear IM problem, with access to the initial state at  $t = 0$ , we repeatedly solve a system matrix  $Y$  (moving forward by  $\Delta t$  in each recursion) following steps described in Fig. 18 to obtain the time-domain response from  $t = 0$  to  $t = t_{final}$ . How we update the system matrix in the nonlinear analysis is different than in the case of linear analysis in Section 1.1.7. Linearized terms in the system matrix  $Y$  corresponding to nonlinear devices are updated *iteratively* after each linear solve. The memory-stamps corresponding to the devices with differential terms are only updated each time we move forward in time (*recursion*). We only move forward in time once the nonlinear stamps have converged (i.e., no considerable difference in the value of linearized stamps in subsequent iterations). As in the linear case, we dynamically adjust  $\Delta t$  at each step based on the trade-off between simulation run-time and local truncation error (LTE).

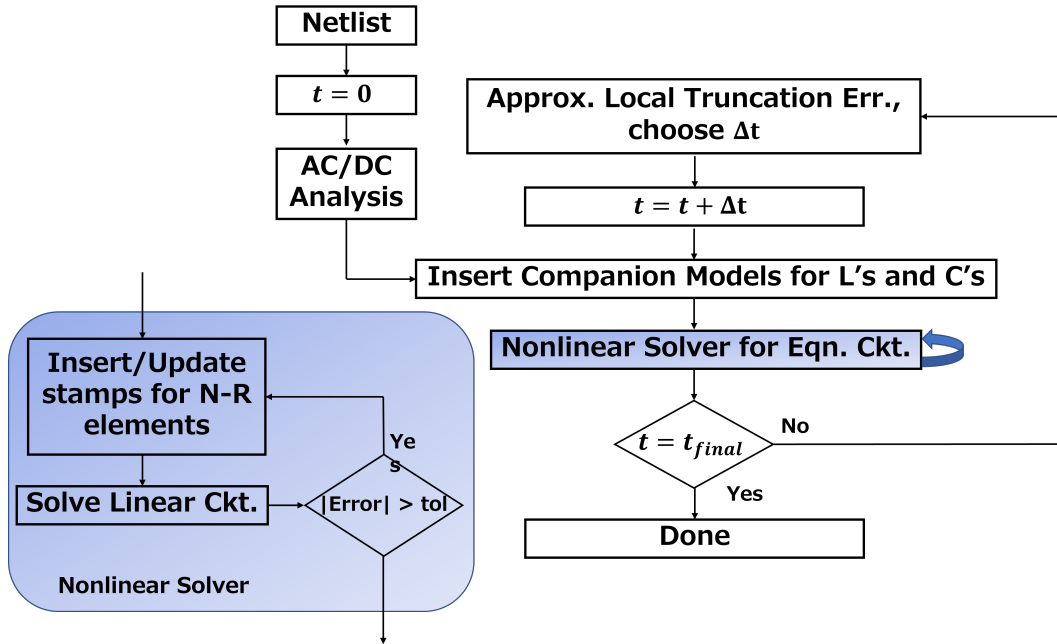


Figure 18: Running EMT simulation for nonlinear networks.

## 2 Acknowledgments

Tim McNamara and Naeem Turner-Bandele have both TA'ed the course where this material was taught and they have spent a significant amount of time ensuring that the derivations and explanations in this document are precise and accurate.

I also want to acknowledge my Ph.D. advisor Larry Pileggi whose book, lecture slides, and guidance were critical in putting this document together.

## References

- [1] L. Pillage, *Electronic Circuit & System Simulation Methods (SRE)*. McGraw-Hill, Inc., 1998.
- [2] C. T. Kelley, *Solving nonlinear equations with Newton's method*. SIAM, 2003.
- [3] P. J. Davis and P. Rabinowitz, *Methods of numerical integration*. Courier Corporation, 2007.
- [4] A. Pandey, M. Jereminov, X. Li, G. Hug, and L. Pileggi, "Unified power system analyses and models using equivalent circuit formulation," in *2016 IEEE Power & Energy Society Innovative Smart Grid Technologies Conference (ISGT)*, IEEE, 2016, pp. 1–5.
- [5] A. Pandey, M. Jereminov, M. R. Wagner, D. M. Bromberg, G. Hug, and L. Pileggi, "Robust power flow and three-phase power flow analyses," *IEEE Transactions on Power Systems*, vol. 34, no. 1, pp. 616–626, 2018.